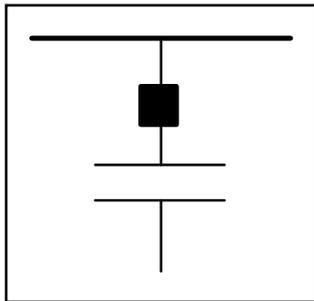
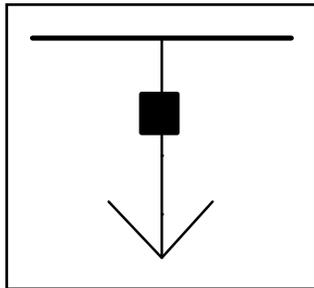
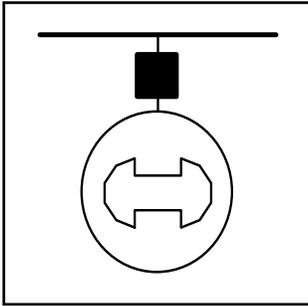
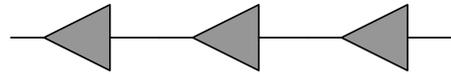


PowerWorld
Corporation



Simulator Version 14



**Interactive power system
simulation, analysis and
visualization**

User's Guide

**2001 South First Street
Champaign, IL 61820
<http://www.powerworld.com>
support@powerworld.com
+1 (217) 384-6330**

Table of Contents

Chapter 1: Getting Started With PowerWorld	1
About this Manual.....	2
Introduction to PowerWorld Simulator	3
Introduction to Simulator Add-On Tools	4
What's New.....	5
Help	11
Contact Information.....	12
Windows Basics.....	13
PowerWorld Simulator: Getting Started.....	16
Edit Mode Introduction.....	17
Run Mode Introduction.....	18
Script Mode Introduction	19
Message Log	20
Custom and Memo Display	21
Status Bar	22
Ribbons User Interface.....	23
Quick Access Toolbar	26
Add Ons Ribbon Tab	27
Case Information Ribbon Tab	30
Draw Ribbon Tab.....	34
Clipboard Ribbon Group	35
Formatting Ribbon Group	37
Individual Insert Ribbon Group	39
Quick Insert Ribbon Group	41
Select Ribbon Group	43
Onelines Ribbon Tab.....	44
Active Ribbon Group	46
Zoom Ribbon Group.....	50
Options Ribbon Tab	51
Tools Ribbon Tab.....	53
Edit Mode Ribbon Group.....	55
Other Tools Ribbon Group	56
Run Mode Ribbon Group	58
Power Flow Tools Ribbon Group.....	59
Window Ribbon Tab.....	62
Chapter 2: Using Online Diagrams	65

Online Diagram Overview	66
Relationship Between Display Objects and the Power System Model.....	67
Online Local Menu.....	68
Online Display Options Dialog	70
Display Options	71
Pie Chart/Gauge Options	73
Animated Flows Options.....	74
Thumbnail View.....	77
Thumb Nail Menu	78
Display Object Options	79
Substation Display Options	81
Online Animation	83
Copying Onelines to Other Programs	84
Display Explorer.....	85
Supplemental Data	87
Movie Maker.....	89
Map Projections.....	90
Great Circle Distance Dialog.....	94
Great Circle Distance Calculation	95
Populate Lon,Lat with Display X,Y	96
Geography/Coordinates.....	97
Shape File Import.....	98
GIS Shapefile Data: Control	99
GIS Shapefile Data: Identify	100
GIS Shapefile Data: Modify.....	101
GIS Shapefile Data: Format	102
GIS Shapefile Data: Shape List.....	103
Shapefile Database Record Dialog	104
Export Oneline As Shapefile.....	105
Insert Measure Line	107
Delete All Measure Lines	108
Path Distances from Bus or Group.....	109
Closet Facilities to Point.....	110
Online Screen Coordinates.....	111
Online Zooming and Panning	112
Save View Level Dialog	114
Window Ribbon Tab.....	116
Online Conditional Display of Objects.....	119
Keyboard Short Cut Actions Dialog	120
Printing Oneline Diagrams	122

Print Options Dialog	123
Printer Setup	125
Chapter 3: Creating, Loading, and Saving Simulator Cases.....	127
Application File Menu	128
Case Formats	129
GE EPC File Load Options	131
Validating a Case	132
Opening a Simulation Case	133
Opening a Oneline Diagram	134
Recently Opened Cases	135
Building a New Case	136
Building a New Oneline	137
Saving Cases	138
Saving a Oneline	139
Close Oneline	140
Auxiliary File Format (*.aux)	141
Generator Capability Curves Format (*.gcp)	142
Generator Cost Data Format (*.gcd)	143
Generator Cost Curve Data Options	144
Identify Cross Compound Units	145
Injection Groups Format (*.inj)	146
Interface Data Format (*.inf)	147
Sequence Data Format	148
Exporting Onelines in Different Graphic Formats	150
Saving Images as Jpegs	151
Saving Admittance Matrix and Jacobian Information	152
Chapter 4: Building a Oneline Diagram.....	153
Edit Mode Overview	154
Edit Mode General Procedures	155
Draw Ribbon Tab	157
Oneline Display Options	158
Anchored Objects	159
Inserting and Placing Multiple Display Fields	160
Area Display Objects	161
Area Display Options Dialog	162
Zone Display Objects	163
Zone Display Options Dialog	164
Super Area Display Objects	165
Super Area Display Options Dialog	166
Owner Display Objects	167

Owner Display Options Dialog	168
Area Fields on Onelines	169
Zone Fields on Onelines	170
Super Area Fields on Onelines.....	172
Owner Fields on Onelines	173
Bus Display Objects	174
Bus Fields on Onelines	175
Old Voltage Gauges	176
Old Voltage Gauge Options Dialog.....	177
Substation Display Objects	178
Substation Fields on Onelines	179
Generator Display Objects	180
Generator Fields on Onelines	181
Load Display Objects.....	182
Load Fields on Onelines.....	183
Transmission Line Display Objects.....	184
Line Fields on Onelines.....	185
Circuit Breakers on Onelines	186
Line Flow Pie Charts on Onelines.....	187
Line Flow Gauges	188
Line Flow Gauge Options Dialog	189
Line Flow Arrows on Onelines	190
DC Transmission Line Display Objects.....	191
Multi-section Line Display Objects	192
Transformer Display Objects.....	193
Transformer Fields on Onelines.....	195
Three Winding Transformer Display Objects.....	196
Series Capacitor Display Objects	198
Series Capacitor Fields on Onelines	199
Switched Shunt Display Objects	200
Switched Shunt Fields on Onelines	201
Interface Display Objects.....	202
Automatically Inserting Interface Display Objects	203
Interface Fields on Onelines.....	204
InterArea Flow Options Dialog	205
Interface Pie Charts on Onelines	206
Loading NERC Flowgates.....	207
Saving NERC Flowgates.....	208
Injection Group Display Objects	209
Links to Onelines and Auxiliary Files	210

Document Links on Onelines.....	212
Background Lines on Onelines	213
Background Lines Dialog	214
Background Rectangles on Onelines	215
Background Ellipses on Onelines	216
Background Pictures on Onelines	217
Converting Background Lines.....	218
Converting Background Ellipses.....	219
Converting Background Rectangles.....	220
Online Fields.....	221
Text on Onelines	222
Memo Text.....	223
Generic Model Fields	224
Supplemental Data Fields on Onelines	225
Pie Charts/Gauges: Lines.....	226
Pie Charts/Gauges: Interfaces.....	228
Pie Charts/Gauges: Pie Chart/Gauge Styles	229
Pie Charts/Gauges: General Options	230
Pie Chart / Gauge Dialogs.....	231
Pie Chart / Gauge Style Dialog	234
Pie Chart / Gauge Style Dialog - Standard Parameters Tab	236
Pie Chart / Gauge Style Dialog - Open Parameters Tab	238
Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab	239
Pie Chart / Gauge Style Dialog - Gauge Parameters Tab.....	240
Pie Chart / Gauge Example.....	242
Chapter 5: Edit Mode Tools and Options.....	243
Appending a Case.....	244
Refresh Anchors	245
Bus Renumbering Dialog	246
Bus Renumbering: Automatic Setup of Bus List Options	247
Bus Renumbering: Bus Change Options	249
Renumber Areas/Zones/Substations Dialog	250
Merging Buses.....	251
Splitting Buses.....	252
Split Bus Dialog	253
Equipment Mover	254
Potential Misplacements Dialog	255
Tapping Transmission Lines.....	256
Automatic Line Tap Dialog.....	257
Setting Default Drawing Options	259

Bus Selection Page	262
List Unlinked Display Objects	264
Set Selected Field	265
Network Cut.....	266
Browse Open Onelines	268
Unused Bus Numbers.....	269
Palette Overview	270
Using the Insert Palettes	271
Automatically Inserting Buses	274
Automatically Inserting Transmission Lines	275
Automatically Inserting Generators.....	277
Automatically Inserting Loads	278
Automatically Inserting Switched Shunts.....	279
Automatically Inserting Interface Display Objects	280
Automatically Inserting Substations	281
Automatically Inserting Borders	282
Equivalents	285
Equivalents Display	286
Scaling.....	289
Overview of Facility Analysis in PowerWorld Simulator	293
Facility Analysis Dialog	294
Augmenting Path Max Flow Min Cut Algorithm	296
Graph Flow	297
Chapter 6: Editing Online Diagrams.....	299
Clipboard Ribbon Group	300
Select by Criteria Dialog	302
Select Ribbon Group	304
Grid/Highlight Unlinked Objects	305
Using the Online Alignment Grid.....	306
Setting Background Color	307
Zoom, Pan, and Find.....	308
Default Drawing Values.....	310
Formatting Ribbon Group	311
Format Multiple Objects	313
Paste Format Dialog.....	314
Font Properties.....	315
Line/Fill Properties	316
Levels/Layers Options	317
Screen Layers.....	318
Screen Layer Options	319

Format Field Properties	320
Display/Size Properties	321
Align Group Objects	322
Clipboard Ribbon Group	323
Undo Command.....	325
Chapter 7: Properties of Simulator Objects	327
Bus Information (Edit Mode).....	328
Bus Field Information.....	331
Shortest Path Between Elements.....	333
Substation Information (Edit Mode).....	334
Substation Field Options.....	336
Generator Information (Edit Mode).....	338
Generator Field Information.....	340
Generator Options: Display.....	342
Generator Options: Power and Voltage Control.....	343
Generator Options: Costs	346
Generator Options: Fault Parameters.....	348
Generator Options: Owners, Area, Zone, Sub	349
Generator Cost Description	350
Set Generator Participation Factors	351
Generator Reactive Power Capability Curve.....	353
Load Information (Edit Mode).....	354
Load Options: Load Information.....	356
Load Options: OPF Load Dispatch	357
Load Field Information	358
Load Modeling.....	360
Line/Transformer Information (Edit Mode)	361
Transmission Line/Transformer Options: Display	363
Transmission Line/Transformer Options: Parameters	364
Line Shunts Information.....	365
Line Per Unit Impedance Calculator Dialog.....	366
Transmission Line Parameter Calculator.....	367
Transmission Line/Transformer Options: Transformer Control.....	377
Transformer AVR Dialog	379
Transmission Line/Transformer Options: Series Capacitor.....	381
Multi-Section Line Information	382
Line Field Options.....	384
Series Capacitor Field Options Dialog.....	386
Transformer Field Options Dialog	387
DC Transmission Line Options.....	389

DC Line Options: Line Parameters	390
DC Line Options: Rectifier Parameters	392
DC Line Options: Inverter Parameters	393
DC Line Options: Actual Flow	394
DC Line Options: OPF.....	395
Multi-Terminal DC Record Information.....	396
Multi-Terminal DC Bus Information.....	397
Multi-Terminal DC Converter Information.....	398
Multi-Terminal DC Line Information.....	400
Transmission Line/Transformer Options: Transformer Control.....	401
Transformers Bases and Impedances Dialog	403
Transformer Impedance Correction Table Display.....	404
Transformer AVR Dialog	405
Transformer Mvar Control Dialog.....	407
Transformer Phase Shifting Information.....	409
Transformer Field Options Dialog	411
Three Winding Transformer Information	413
Switched Shunt Information (Edit Mode).....	415
Switched Shunt Field Information.....	419
Zone Information (Edit Mode).....	421
Bus Information (Run Mode)	423
Substation Information (Run Mode).....	426
Generator Information (Run Mode)	428
Generator Information: Power and Voltage Control.....	430
Generator Options: Costs	433
Generator Information: OPF	435
Generator Options: Owners, Area, Zone, Sub	436
Load Information (Run Mode).....	437
Line/Transformer Information (Run Mode).....	439
Transformer AVR Dialog	443
Transformer Mvar Control Dialog.....	445
Transformer Phase Shifting Information.....	447
Transformer Impedance Correction Tables Dialog.....	449
Switched Shunt Information (Run Mode)	450
Zone Information (Run Mode).....	454
Labels.....	456
Label Manager Dialog.....	459
Area Information	460
Area Information: Buses.....	462
Area Information: Gens.....	463

Area Information: Loads.....	464
Area Information: Area MW Control Options.....	465
Area Information: Info/Interchange	466
Area Information: Options	467
Area Information: Tie Lines	468
Area Information: OPF	469
Area Information: Custom	470
Area Field Options	471
Super Area Information Dialog.....	473
Super Area Field Information	476
Interface Information.....	478
Interface Element Information	480
Interface Field Information Dialog	481
Interface Pie Chart Information Dialog.....	482
Automatically Inserting Interfaces in Case.....	483
Nomogram Information Dialog.....	484
Injection Group Overview	486
Creating Injection Groups.....	487
Auto Insert Injection Groups.....	488
Deleting Injection Groups.....	490
Injection Group Display	491
Injection Group Dialog	493
Import PTI Subsystems Dialog.....	495
Participation Points Overview	496
Participation Point Records Display.....	497
Add Participation Points Dialog.....	500
Chapter 8: Case Information Displays	503
Model Explorer	504
Case Information Displays	507
Configuring the Case Information Displays	508
Custom Field Descriptions.....	512
Custom Field Description Dialog	513
Calculated Fields.....	515
Case Information Customizations Display	517
Case Information Displays: Local Menu Options.....	518
Case Information Displays: Colors and Cell Styles	520
Case Information Displays: Using Cell Handles	521
Case Information Displays: Sorting Records	522
Case Information Displays: Finding Records	523
Saving Case Information Display Contents as HTML Tables	524

HTML Table Format Dialog	525
Case Information Toolbar	526
Case Information Toolbar: Copy, Paste and Send Menu	529
Case Information Toolbar: Filtering Menu	531
Case Information Toolbar: Geo Data View Menu.....	533
Case Information Toolbar: Load Auxiliary Files Menu	534
Case Information Toolbar: Save Auxiliary Files Menu	535
Case Information Toolbar: Records Menu	537
Case Information Toolbar: Set, Toggle, and Columns Menus	539
Area/Zone/Owner Filters	541
Advanced Filtering	542
Advanced Filters Dialog	543
Advanced Filtering: Advanced	544
Advanced Filtering: Device	546
Advanced Filters Display.....	547
Custom Expressions	548
Custom Expressions Display.....	549
Functions and Operators Available.....	550
Find Dialog Basics.....	553
Search for Text Dialog.....	555
Model Conditions Display and Dialog	556
Model Filters Display and Dialog.....	557
Entering a Range of Numbers	558
Copying Simulator Data to and from Other Applications	559
Save Case Information Data	560
Key Fields	561
Required Fields.....	563
Contour Column Dialog	564
Contour Column Type	565
Grid Metrics Dialog.....	567
Auxiliary File Export Format Description (For both Display and Power System) ...	568
Geographic Data View.....	572
Geographic Data View Styles: Fields and Attributes.....	574
Geographic Data View Styles: General Display Options.....	578
Geographic Data View Styles	579
Custom Case Information Display.....	581
Define Fields/Strings.....	582
Change Field Data	583
Show Fields Primary.....	584
Show Fields Secondary.....	586

Custom Case Information Display Local Menu	587
User-Defined Case Information Displays	588
Case Description.....	590
Case Summary	591
Power Flow List	593
Quick Power Flow List	595
Outages.....	596
Area Display	597
Zone Display	599
Tie Lines between Areas Display	600
Tie Lines between Zones Display.....	602
Super Area Display	603
Bus Display	605
Remotely Regulated Bus Display	606
Bus Mismatches Display	608
Substation Records Display.....	609
Generator Display	610
Generator/Load Cost Models.....	612
Generator Cost Models Display.....	613
Generator Cubic Cost Display.....	615
Generator Piecewise Linear Cost Display	617
Load Display	619
Load Benefit Models Display	620
Line and Transformer Display	622
Multi-Section Lines Display	624
Transformer Display.....	626
Three Winding Transformer Display.....	628
Line Shunts Display.....	629
DC Lines Display.....	630
Multi-Terminal DC Record Display	631
Switched Shunt Display	632
Interface Display	634
Nomogram Display	636
Injection Group Display	637
Injection Group Dialog	639
Participation Point Records Display	641
Island Display.....	644
MW Transactions Display.....	645
Transaction Dialog	646
Limit Monitoring Settings.....	648

Limit Monitoring Settings and Limit Violations Dialog.....	649
Limit Group Dialog.....	654
Owner Data Information Display.....	656
Owner Dialog.....	657
Owned Bus Records Display.....	658
Owned Load Records Display.....	659
Owned Generator Records Display.....	660
Owned Line Records Display.....	661
Jacobian Display.....	662
Ybus Display.....	663
Chapter 9: Solving and Simulating a Case.....	665
Simulator Options.....	666
Power Flow Solution Options.....	667
Power Flow Solution: Common Options.....	668
Power Flow Solution: Advanced Options.....	671
Post Power Flow Solution Actions Dialog.....	674
Power Flow Solution: Island-Based AGC.....	676
Power Flow Solution: DC Options.....	678
DC Power Flow Loss Setup.....	680
Power Flow Solution: General.....	681
Power Flow Solution: Storage.....	682
Message Log Options.....	683
Environment Options.....	685
Online Options.....	687
Case Information Display Options.....	689
Limits Options.....	692
File Management Options.....	693
Power Flow Tools Ribbon Group.....	695
Solving the Power Flow.....	698
Area Control.....	701
Set Generator Participation Factors.....	703
Transaction Dialog.....	705
Calculate MW-Distance.....	707
MW-Distance Options.....	708
Area Control Error (ACE) Chart.....	709
Area Load and Generation Chart.....	710
Area Losses Chart.....	711
Area MW Transactions Chart.....	712
Area Average Cost Chart.....	713
Bus View Display.....	714

Substation View Display	717
Chapter 10: Run Mode Tools and Options.....	719
Generator Economic Curves.....	720
Find Branches that Create Islands	721
Custom Hint Values.....	722
Line Loading Replicator	724
Dynamic Formatting Overview	727
Dynamic Formatting Dialog.....	728
Difference Flows.....	730
Difference Flows: Case Types	731
Difference Flows Dialog.....	732
Using Difference Flows.....	733
Present Topological Differences from Base Case.....	735
Governor Power Flow.....	737
Governor Power Flow: Generator Options (Ignore Area/Zone/Owner filter) Tab..	738
Governor Power Flow: Options Tab	739
Contingency Analysis: An Introduction.....	740
Available Contingency Actions.....	741
Contingency Analysis: Terms	743
Contingency Solution Options Dialog	744
Contingency Case References	745
Contingency Case References - State Information.....	746
Contingency Case References - Defining the Reference State	748
Contingency Case References - Reference State Solution Options	749
Contingency Analysis: Defining Contingencies	750
Auto Insert Contingencies.....	751
Loading Contingencies from a File	754
PSS/E Contingency Format	755
PSS/E Load Throw Over Files.....	756
Saving Contingency Records to a File	757
Global Actions	759
Contingency Blocks.....	760
Contingency Analysis Dialog Overview	762
Running the Contingency Analysis.....	763
Other Contingency Actions	764
Contingency Analysis Dialog - Contingencies Tab.....	766
Contingency Violations Display.....	769
Contingency Definition Display.....	771
Lines, Buses and Interfaces Tab.....	773
Lines, Buses and Interfaces Tab - Contingencies Section.....	774

Contingency Definition Display	775
Contingency Options Tab.....	777
Contingency Options Tab: Modeling	778
Contingency Options Tab: Limit Monitoring	780
Contingency Options Tab: Advanced Modeling	783
Model Expressions	786
Bus Load Throw Over Records	789
Contingency Options Tab: Report Writing	790
Contingency Options Tab: Miscellaneous.....	792
Contingency Summary Tab	793
Comparing Contingency Analysis Results.....	794
Comparing Contingencies List Displays	796
Contingency Definition Dialog	799
Contingency Element Dialog	802
Make-Up Power Sources.....	813
Contingency Make-Up Sources Dialog	814
Tutorial: Contingency Analysis	815
Tutorial: Contingency Analysis - Page 2	816
Tutorial: Contingency Analysis - Page 3	819
Tutorial: Contingency Analysis - Page 4	821
Tutorial: Contingency Analysis - Page 5	823
Tutorial: Contingency Analysis - Page 6	824
Tutorial: Contingency Analysis - Page 7	826
Tutorial: Contingency Analysis - Page 8	827
Tutorial: Contingency Analysis - Page 9	828
Tutorial: Contingency Analysis - Page 10	829
Tutorial: Contingency Analysis - Page 11	830
Fault Analysis	831
Fault Analysis Dialog	832
Fault Analysis Bus Records	835
Fault Analysis Generator Records	836
Fault Analysis Line Records.....	837
Mutual Impedance Records	838
Mutual Impedance Record Dialog	839
Fault Analysis Load Records.....	840
Fault Analysis Switched Shunt Records	841
Contouring	842
Contouring Options.....	843
Contour Type	844
Contour Type Options	848

Custom Color Map	850
Functional Description of Contour Options.....	852
Power Transfer Distribution Factors.....	856
Power Transfer Distribution Factors Dialog.....	857
Line Outage Distribution Factors (LODFs)	861
Line Outage Distribution Factors Dialog	863
Advanced LODF Calculation Dialog	865
Directions Display	866
Directions Dialog	867
Auto Insert Directions	868
Flow and Voltage Sensitivities	869
Loss Sensitivities.....	871
Transmission Loading Relief Sensitivities.....	874
TLR Sensitivities Dialog	875
TLR Multiple Device Type	878
Generation Shift Factor Sensitivities.....	880
Chapter 11: Time Step Simulation	881
Time Step Simulation	882
Time Step Simulation: Quick Start.....	883
Time Step Simulation Dialog	885
Time Step Simulation Toolbar	888
Time Step Simulation Pages.....	889
Hourly Summary Page.....	890
Hourly Summary Page: Local Menu	892
Input Page.....	893
Matrix Grids	894
Results: Constraints Page.....	895
Binding Elements Dialog.....	898
Results Page	899
Custom Results Selection Dialog.....	900
Results Grid Pages.....	901
Time Step Simulation Options.....	902
Specifying and Maintaining a List of Timepoints.....	905
New Timepoint Dialog.....	906
Change Timepoint Time Dialog	907
TSB Case Description Page	908
Insert/Scale Column Dialog.....	909
Loading Hourly Input Data	911
Setting up Scheduled Input Data.....	912
Schedule Dialog.....	913

Schedules Page	915
Schedule Subscription Dialog	916
Schedule Subscriptions Page.....	918
Running a Timed Simulation	920
Running OPF and SCOPF Time Step Simulations	921
Storing Input Data and Results	924
Chapter 12: PowerWorld Simulator Add-On Tools.....	925
Introduction to Simulator Add-On Tools	926
PowerWorld Simulator PV/QV Overview	927
PV Curves	928
PV Curves Dialog	929
PV Curves: Setup.....	931
PV Curves Setup: Common Options.....	932
PV Curves Setup: Advanced Options.....	935
PV/QV Quantities to Track	940
PV Curves: Limit Violations	944
PV Curves: Output	946
PV Curves: QV Setup	949
PV Curves: PV Results.....	952
PV Curves Results: Overview	955
PV/QV Curves Results: Plot.....	958
PV/QV Curves Results: Track Limits.....	960
QV Curves.....	961
QV Curves: Buses	963
QV Curves: Options	965
QV Curves Options: Solution.....	966
QV Curves Options: Contingencies	968
QV Curves Options: Output	970
PV/QV Quantities to Track.....	974
QV Curves: Results	978
QV Curves Results: Listing.....	979
PV/QV Curves Results: Plot.....	981
PV/QV Curves Results: Track Limits.....	983
PV/QV Refine Model.....	984
PowerWorld Simulator Optimal Power Flow Overview.....	986
OPF Objective Function	987
OPF Equality and Inequality Constraints.....	988
OPF Equality Constraints	989
OPF Inequality Constraints	990
Determining Set of Active Inequality Constraints	992

OPF Unenforceable Constraints	993
OPF Marginal Costs	994
OPF Primal LP	995
OPF Future Enhancements	996
OPF Options and Results	997
OPF Options: Common Options.....	998
OPF Options: Constraint Options.....	1000
OPF Options: Control Options	1002
OPF Options: Advanced Options.....	1004
OPF Options: Solution Summary	1005
OPF Options: All LP Variables.....	1008
OPF Options: LP Basic Variables.....	1009
OPF Options: LP Basis Matrix	1010
OPF Options: Bus MW Marginal Price Details	1011
OPF Options: Bus MVAR Marginal Price Details	1012
OPF Options: Bus Marginal Controls	1013
OPF Options: Inverse of LP Basis	1014
OPF Area Records.....	1015
OPF Bus Records	1017
OPF Generator Records	1018
OPF Interface Records	1021
OPF Nomogram Records	1022
OPF Line/Transformer Records	1023
OPF DC Lines Records.....	1024
OPF Load Records	1025
OPF Super Area Records	1027
OPF Controls	1029
OPF Phase Shifter Records	1030
OPF Example - Introduction.....	1031
OPF Example - Marginal Costs	1034
OPF Example - Super Areas	1035
OPF Example - Enforcing Line MVA Constraints	1037
Security Constrained Optimal Power Flow Overview	1039
SCOPF Objective Function	1040
SCOPF Dialog	1041
SCOPF Solution Process.....	1042
SCOPF Results.....	1043
SCOPF Equality and Inequality Constraints.....	1044
SCOPF Equality Constraints.....	1045
SCOPF Inequality Constraints.....	1046

SCOPF Control.....	1048
SCOPF CTG Violations	1050
SCOPF LP Solution Details	1052
SCOPF All LP Variables	1053
SCOPF LP Basic Variables	1055
SCOPF LP Basis Matrix	1056
SCOPF Bus Marginal Price Details.....	1057
SCOPF Bus Marginal Controls.....	1058
SCOPF Example: Introduction	1059
SCOPF Example: Marginal Prices	1062
SCOPF Example: Unenforceable Constraints.....	1063
Optimal Power Flow Reserves Overview.....	1064
OPF Reserves Topics.....	1065
OPF Reserves Controls	1066
Generator and Load OPF Reserves Bids	1067
OPF Reserves Constraints.....	1068
Area and Zone OPF Reserve Requirement Curves	1069
OPF Reserves Objective Function	1070
OPF Reserves Results and Pricing.....	1071
OPF Reserves Case Information Display.....	1072
OPF Reserves Example.....	1073
Available Transfer Capability (ATC) Analysis	1077
Available Transfer Capability Dialog.....	1078
ATC Dialog: Options.....	1080
ATC Dialog Options: Common Options.....	1081
ATC Dialog Common Options: Transfer Result Reporting Options.....	1085
ATC Dialog Options: Advanced Options.....	1088
ATC Extra Monitors Dialog	1093
ATC Dialog: Result.....	1095
ATC Dialog: Analysis	1096
Multiple Scenario Available Transfer Capability Dialog	1098
Multiple Scenario ATC Dialog: Scenarios	1100
Multiple Scenario ATC Dialog: Results.....	1103
Multiple Scenario ATC Analysis Results: Local Menu Options.....	1105
Multiple Scenario ATC Dialog: Combined Results	1106
ATC Dialog: Transfer Limiters Display	1108
ATC Analysis Methods: Solution Methods	1112
ATC Analysis Methods: Single Linear Step (SL).....	1113
ATC Analysis Methods: Iterated Linear Step (IL).....	1114
ATC Analysis Methods: Iterated Linear Step (IL) then Full CTG Solution.....	1117

Simulator Automation Server (SimAuto) Overview	1121
Installing Simulator Automation Server	1122
Including Simulator Automation Server Functions	1123
Connecting to Simulator Automation Server	1124
Passing Data to the Simulator Automation Server	1126
Getting Data from the Simulator Automation Server.....	1127
Simulator Automation Server Functions	1128
ChangeParameters Function	1129
ChangeParametersSingleElement Function	1130
ChangeParametersSingleElement Sample Code	1131
ChangeParametersMultipleElement Function	1133
ChangeParametersMultipleElement Sample Code.....	1134
ChangeParametersMultipleElementFlatInput Function	1136
ChangeParametersMultipleElementFlatInput Sample Code.....	1137
CloseCase Function	1139
CloseCase Function: Sample Code	1140
GetFieldList Function.....	1141
GetFieldList Function: Sample Code.....	1142
GetParametersSingleElement Function.....	1143
GetParametersSingleElement Function: Sample Code Borland® Delphi.....	1144
GetParametersSingleElement Function: Sample Code Matlab®	1146
GetParametersSingleElement Function: Sample Code Microsoft® Visual Basic for Excel	1148
GetParametersMultipleElement Function	1150
GetParametersMultipleElement Sample Code Borland® Delphi.....	1152
GetParametersMultipleElement Function: Sample Code Matlab®.....	1153
GetParametersMultipleElement Function: Sample Code Microsoft® Visual Basic for Excel	1155
GetParametersMultipleElementFlatOutput Function	1157
GetParameters Function	1158
ListOfDevices Function	1159
ListOfDevices Function: Sample Code for Borland® Delphi	1161
ListOfDevices Function: Sample Code for Matlab®	1163
ListOfDevices Function: Sample Code for Microsoft® Visual Basic for Excel.....	1165
ListOfDevicesAsVariantStrings Function.....	1167
ListOfDevicesFlatOutput Function	1168
LoadState Function	1169
LoadState Function: Sample Code	1170
OpenCase Function.....	1171
OpenCase Function: Sample Code.....	1172
ProcessAuxFile Function	1173
ProcessAuxFile Function: Sample Code	1174

RunScriptCommand Function	1175
RunScriptCommand Function: Sample Code	1176
SaveCase Function.....	1177
SaveCase Function: Sample Code.....	1178
SaveState Function	1179
SaveState Function: Sample Code	1180
SendToExcel Function	1181
SendToExcel Function: Sample Code	1182
WriteAuxFile Function	1183
WriteAuxFile Function: Sample Code	1184
Simulator Automation Server Properties.....	1186
ExcelApp Property.....	1187
ExcelApp Property: Sample Code.....	1188
CurrentDir Property.....	1189
CurrentDir Property: Sample Code.....	1190
ProcessID Property	1191
ProcessID Property: Sample Code	1192
RequestBuildDate Property	1193
PowerWorld Object Variables.....	1194
Simulator Automation Server (version 9).....	1195
Installing Simulator Automation Server (version 9)	1196
Including Simulator Automation Server Functions (version 9)	1197
Connecting to Simulator Automation Server (version 9).....	1198
Simulator Automation Server Properties (version 9)	1200
ExcelApp Property (version 9).....	1201
Simulator Automation Server Functions (version 9).....	1202
ChangeParameters Function (version 9).....	1203
CloseCase Function (version 9)	1204
GetParameters Function (version 9)	1205
ListOfDevices Function (version 9)	1206
LoadContingencies Function (version 9).....	1207
OpenCase Function (version 9)	1208
ProcessAuxFile Function (version 9).....	1209
RunScriptCommand Function (version 9)	1210
SaveCase Function (version 9).....	1211
SendToExcel Function (version 9)	1212
WriteAuxFile Function (version 9).....	1213
PowerWorld Object Variables (Version 9).....	1214
Topology Processing Overview.....	1215
Topology Processing: Full-Topology Model	1217

Topology Processing: Superbuses	1218
Topology Processing: Subnets	1219
Topology Processing: Consolidation.....	1220
Topology Processing Consolidation: Primary Bus Priority Scheme	1221
Topology Processing Consolidation: Illustration of Process.....	1222
Topology Processing: Consolidated Representation.....	1223
Topology Processing: Consolidated Superbus View	1224
Topology Processing Dialog	1226
Topology Processing: Power Flow Using Topology Processing	1228
Topology Processing: Ill-Conditioned Jacobian.....	1229
Topology Processing: Contingency Analysis Using Topology Processing	1230
Topology Processing: Incremental Topology Processing	1231
Topology Processing: Topology Processing in Applications.....	1232
Topology Processing: Saving the Planning Case.....	1233
Topology Processing: Planning Cases	1234
Topology Processing: Primary Bus Mapping.....	1235
Topology Processing: Breaker-to-Device Contingency Conversion.....	1237
Transient Stability Overview	1238
Transient Stability Case Info Menu	1239
Transient Stability Analysis Dialog	1241
Transient Stability Analysis: Simulation Control.....	1242
Transient Stability Analysis: Options	1244
Transient Stability Analysis Options: General	1245
Transient Stability Analysis Options: Power System Model	1247
Transient Stability Analysis: Results to Save	1249
Transient Stability Analysis: Plots	1252
Transient Stability Analysis Plots: Plot Designer	1256
Transient Stability Analysis Plot Designer: Plot.....	1260
Transient Stability Analysis Plot Designer: Chart	1262
Transient Stability Analysis Plot Designer: Horizontal Axis.....	1264
Transient Stability Analysis Plot Designer: Vertical Axis.....	1266
Transient Stability Analysis Plot Designer: Plot Series List.....	1268
Transient Stability Analysis Plot Designer: Plot Series.....	1269
Transient Stability Analysis: Results.....	1271
Transient Stability Analysis: States/Manual Control	1273
Transient Stability Analysis: Validation	1275
Chapter 13: Auxiliary Script/Data Files	1277
Auxiliary Files	1278
Script Command Execution Dialog	1280
Quick Auxiliary Files Dialog	1281

Script Section	1282
Script General Actions	1284
Script Online Actions	1297
Script Edit Mode Actions.....	1299
Script Run Mode Actions.....	1304
Script PowerFlow Submode Actions	1312
Script Contingency Submode Actions	1316
Script ATC Submode Actions.....	1318
Script Fault Submode Actions.....	1320
Script PV Submode Actions.....	1321
Script QV Submode Actions	1323
Data Section	1325
Data Argument List.....	1326
Data Key Fields	1329
Data List.....	1330
Data SubData Sections.....	1331
Data ATC_Options	1332
Data ATCExtraMonitor.....	1333
Data ATCScenario.....	1334
Data AuxFileExportFormatData	1336
Data AuxFileExportFormatDisplay.....	1337
Data BGCALculatedField	1338
Data Bus.....	1339
Data BusViewFormOptions	1340
Data ColorMap	1341
Data Contingency	1342
Data CTG Options.....	1348
Data CTGElementBlock.....	1349
Data CustomCaseInfo	1350
Data CustomColors	1351
Data DataGrid	1352
Data DynamicFormatting.....	1353
Data Filter.....	1355
Data Gen	1357
Data GeoDataViewStyle.....	1358
Data GlobalContingencyActions.....	1360
Data HintDefValues.....	1361
Data InjectionGroup.....	1362
Data Interface.....	1363
Data LimitSet	1365

Data Load	1366
Data LPVariable	1367
Data ModelCondition	1368
Data ModelExpression	1369
Data ModelFilter	1370
Data MTDCRecord	1371
Data MultiSectionLine.....	1375
Data Nomogram	1376
Data NomogramInterface	1377
Data Owner	1378
Data PieChartGaugeStyle.....	1380
Data PostPowerFlowActions	1381
Data PWCaseInformation	1382
Data PWFormOptions.....	1383
Data PWLPOPFGTGViol.....	1384
Data PWLPTabRow	1385
Data PWPVResultListContainer	1386
Data QVCurve	1387
Data QVCurve_Options.....	1388
Data SelectByCriteriaSet.....	1389
Data ShapeFileExportDescription.....	1391
Data StudyMWTransactions	1392
Data SuperArea	1393
Data TSSchedule	1394
Data UserDefinedDataGrid.....	1395
Data View	1396
Data Section for Display Auxiliary File	1397
Data Argument List for Display Auxiliary File	1398
Data Key Fields for Display Auxiliary File	1400
Data List for Display Auxiliary File.....	1401
Special Data Sections	1402
Data Subdata Sections for Display Auxiliary File.....	1403
Data Common Sections	1404
Data DisplayLine.....	1405
Data PieChartGaugeStyle.....	1406
Data View	1407
Chapter 14: PowerWorld Simulator Project Files	1409
Overview of PowerWorld Simulator Project Files	1410
PowerWorld Project Initialization Script.....	1411
Associating Project Files with Simulator	1412

Creating a New Project File.....	1413
Create Project Dialog	1414
Opening an Existing Project.....	1416
Chapter 15: Tutorials	1417
Tutorial: Creating a New Case Page 1 of 13	1418
Tutorial: Inserting a Bus Page 2 of 13	1419
Tutorial: Inserting a Generator Page 3 of 13	1421
Tutorial: Saving the Case Page 4 of 13	1423
Tutorial: Entering a Second Bus with Load Page 5 of 13	1424
Tutorial: Inserting a Transmission Line Page 6 of 13.....	1427
Tutorial: Inserting a Line Flow Pie Chart on a Transmission Line Page 7 of 13.....	1429
Tutorial: Inserting Circuit Breakers Page 8 of 13.....	1431
Tutorial: Inserting a Transformer Page 9 of 13.....	1433
Tutorial: Inserting a Switched Shunt Page 10 of 13	1436
Tutorial: Inserting Text, Bus and Line Fields Page 11 of 13	1438
Tutorial: Solving the Case Page 12 of 13	1442
Tutorial: Adding a New Area Page 13 of 13.....	1444
Starting with an Existing Case Page 1 of 15	1447
Tutorial: Loading an Existing Power Flow File Page 2 of 15	1448
Tutorial: Case Information Displays Page 3 of 15	1449
Tutorial: Solving the Case Page 4 of 15	1450
Tutorial: Building Onelines Page 5 of 15	1451
Tutorial: Entering a Bus Page 6 of 15	1452
Tutorial: Automatic Line Insertion Page 7 of 15.....	1453
Panning and Zooming Page 8 of 15.....	1454
Tutorial: Adding Background Page 9 of 15	1455
Tutorial: Simulating the Case Page 10 of 15.....	1456
Tutorial: Run-time Object Dialogs Page 11 of 15	1457
Online Local Menu Page 12 of 15	1458
Tutorial: Area Page 13 of 15.....	1459
Limit Violations Page 14 of 15.....	1460
Other Case Information Displays Page 15 of 15.....	1461
Tutorial: Solving an OPF Page 1 of 6	1462
Tutorial: OPF Three Bus Example Page 2 of 6	1463
Tutorial: OPF Line Limit Enforcement Page 3 of 6	1464
Tutorial: OPF LMP Explanation Page 4 of 6.....	1465
Tutorial: OPF Marginal Cost of Enforcing Constraints Page 5 of 6	1466
Tutorial: OPF Unenforceable Constraints Page 6 of 6	1467
Tutorial: Contingency Analysis	1469

Chapter 1: Getting Started With PowerWorld

This chapter provides the essential information you need to start using PowerWorld Simulator.

The following material is included:

- About this Manual
- Introduction to PowerWorld Simulator
- Introduction to Simulator Add-on Tools
- What's New in Version 14
- Getting Help
- PowerWorld Interface
- Toolbars

About this Manual

PowerWorld Simulator includes comprehensive, context-sensitive on-line help in addition to a printed manual. Using the on-line help is strongly recommended, since it provides features not available in a printed manual, such as the ability to jump to another topic. Furthermore, the on-line help is likely to be the more up-to-date reference as program updates are issued.

Introduction to PowerWorld Simulator

PowerWorld® Simulator (Simulator) is a power system simulation package designed from the ground up to be user-friendly and highly interactive. Simulator has the power for serious engineering analysis, but it is also so interactive and graphical that it can be used to explain power system operations to non-technical audiences. With [Version 14](#) we've made Simulator more powerful, more visual, and easier to use.

Simulator consists of a number of integrated products. At its core is a comprehensive, robust Power Flow Solution engine capable of efficiently solving systems of up to 100,000 buses. This makes Simulator quite useful as a stand-alone power flow analysis package. Unlike other commercially available power flow packages, however, Simulator allows the user to visualize the system through the use of full-color [animated online diagrams](#) complete with [zooming and panning](#) capability. System models can be either modified on the fly or built from scratch using Simulator's full-featured graphical case editor. Transmission lines can be switched in (or out) of service, new transmission or generation can be added, and new transactions can be established, all with a few mouse clicks. Simulator's extensive use of graphics and animation greatly increases the user's understanding of system characteristics, problems, and constraints, as well as of how to remedy them.

The base package of Simulator is capable of solving power systems comprised of up to 100,000 buses. The base package also contains all the tools necessary to perform integrated economic dispatch, area transaction economic analysis, power transfer distribution factor (PTDF) computation, short circuit analysis, and contingency analysis. All of the above features and tools are easily accessible through a consistent and colorful visual interface. These features are so well integrated that you will be up and running within minutes of installation.

In addition to the features of the base Simulator package, various add-on tools are available. Please see [Introduction to Simulator Add-On Tools](#) for more information.

Introduction to Simulator Add-On Tools

In addition to the features of the base Simulator package, various add-on tools are available. A brief introduction to each follows:

Voltage Adequacy and Stability Tool (PVQV)

The purpose of the PVQV add-on is to allow the user to analyze the voltage stability characteristics of a system. After the PVQV simulation is complete, the user can graph various system parameters. For more information, see the [PVQV Overview](#).

Optimal Power Flow Tool (OPF)

The purpose of an OPF is to minimize an [objective \(or cost\) function](#). In Simulator OPF the Linear Programming OPF algorithm (LP OPF) determines the optimal solution by iterating between solving a standard power flow and solving a linear program to change the system controls thereby removing any limit violations. For more information, see the [OPF Overview](#).

Security Constrained Optimal Power Flow Tool (SCOPF)

The OPF tool minimizes an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. The SCOPF tool takes it one step further by considering contingencies that may arise during system operation and ensuring that in addition to minimizing the objective function, no unmanageable contingency violations occur. For more information, see the [SCOPF Overview](#).

Optimal Power Flow Reserves (OPFR)

Simulator OPF Reserves is the tool used to simulate Ancillary Services Reserve Markets. For more information, see the [Optimal Power Flow Reserves Overview](#).

Available Transfer Capability Analysis Tool (ATC)

ATC analysis determines the maximum MW transfer possible between two parts of a power system without violating any limits. For more information, see the [ATC Analysis Overview](#).

PowerWorld Simulator Automation Server (SimAuto)

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on SimAuto, see the [SimAuto Overview](#).

Topology Processing

Topology Processing allows you to solve EMS-type, full-topology models in a numerically robust and transparent manner. Topology Processing extends Simulator applications traditionally used by planners so they can operate in an operations real-time environment. For more information, see the [Topology Processing Overview](#).

Transient Stability (TS)

Transient Stability analysis allows the analysis of system dynamic response to a fault on the system. For more information, see the [Transient Stability Overview](#).

What's New

Web links to previous version information: [Version 13](#) [Version 12](#) [Version 11](#) [Version 10](#) [Version 9](#) [Version 8](#) [Version 7](#) [Version 6](#)

Simulator Version 14 contains a number of major new features and several smaller enhancements designed to improve the performance and convenience of the package.

- New [Topology Processing](#) Add-On
- New [Transient Stability](#) Add-On
- User Interface
 - Added ability to make a specific window Free-Floating or In the Container Window. Most dialogs default as in the Container Window, but the message log defaults as Free-Floating on Top. See the [Window Basics](#) help for changing the window style.
 - As various dialogs are opened, PowerWorld will keep track of the Free-Floating/Contained style as well as the position and size of each dialog. When that dialog is then opened again it will open to the same style, size, and position as it was previously. These settings are stored in the Windows registry so that they are remembered when the program is opened again. See the [Window Basics](#) help for more information.
 - When solving the power flow, **Red**, **Purple**, **Green**, and **Blue** outlines have been added around [Message Log](#) text to signify which part of the power flow solution the message text is originating from (see information in [Solving the Power Flow](#) regarding the Power Flow, Controller, and MW Control loops).
 - In the [Message Log](#), message text which originates from the PV, QV, OPF, or SCOPF calculations will be highlighted in yellow. Log messages which originate from contingency analysis events are highlighted in light blue. For difficult solutions, this highlighting helps determine which part of the solution is having trouble.
- Advanced Filtering Changes
 - The [Case Information Toolbar: Filtering Menu](#) Filter List drop-down menu has been modified to place the list of filters in a submenu if there are more than 10 filters. Additionally an active advanced filter is denoted with a check mark and the object type of that advanced filter is denoted with a gray check mark.
 - Added support on the list of Advanced Filters display for deleting and inserting advanced filters directly from the [Advanced Filters Case Information display](#).
 - Added [Advanced Filtering by Device](#). Device Filtering allows you to directly use one of the power system model objects as a filter. The relationship between the object type being listed and the object type of the device filter determines how filtering is done.
 - Added the [Case Information Filter bar](#) that provides much more direct access to the advanced and device filtering abilities.
- Auxiliary Files and Display Auxiliary Files
 - Added an optional parameter in the [header of a DATA section](#) that specifies whether to create a new object if an existing one is not found. If omitted the default behavior of prompting the user will continue. If the value is YES, objects will be created, if the value is NO, objects will not be created.
 - When saving out the [limit monitoring settings](#), previously no information regarding specific voltage limits for specific buses was stored. This is now automatically added to the Limit Monitoring Auxiliary file when saved (this includes on the Limit Monitoring dialog as well as from the Contingency analysis, ATC, PV, QV, etc... AUX files).
 - On the [Auxiliary File Export Format Description](#) dialog, a button has been added to Create Format for Complete Case. This automatically creates a format description that is similar to that used by Simulator when choosing to Save Case, As Auxiliary File.
- Auxiliary File SCRIPT and SimAuto Changes

- o Previously, when starting a SimAuto process, an existing Simulator instance would be taken over by the SimAuto process. This has been changed so that [existing Simulator instances are left open](#) and a new Simulator process is started by SimAuto.
- o New and modified script commands
 - Added a new script command [CloseOnline\("filename"\)](#)
 - Added a new script command when running SimAuto called [LoadAXD\("AXDfilename", "onlinefilename", CreateIfNotFound\)](#)
 - Added a new script command [CalculateLODFMatrix\(WhichOnes, filterProcess, filterMonitor, MonitorOnlyClosed, LinearMethod\)](#)
 - Added a new script command [SaveObjectFields\("filename", objecttype, \[fieldlist\]\)](#)
 - Added a new script command [ChangeSystemMVABase\(NewBase\)](#)
 - Added a new script command [SaveOnline\("filename", "online", savefiletype\)](#)
- Available Transfer Capability (ATC) Enhancements
 - o Added optional parameter [AppendFile to ATCWriteResultsAndOptions](#). YES means results will be appended to an existing file. NO means existing file will be overwritten. Default is YES if not specified to retain previous functionality.
 - o When running iterated ATC methods, ramping transfer, or moving to one of the multiple scenarios, the [minimum voltages for reducing load are set to 0](#).
 - o Added the ability to [interact using Auxiliary files and SimAuto with the ATC Scenario Transfer Limitations by using the objecttype "TransferLimit"](#). This is done by using fields ATCLineZoneChanges, ATCGenChanges, and ATCInterfaceChanges as integer identifier to specify the ATC Scenario the transfer limitations corresponds to.
 - o Added a new tab for multiple ATC Scenario results called ["Combined Results"](#). This shows a single case information display showing transfer limitations under ALL scenarios. Default columns for the fields ATCLineZoneChanges, ATCGenChanges, and ATCInterfaceChanges are included.
 - o Added script command [ATCWriteToText\("filename", filetype\)](#) where filetype can be TAB or CSV. If omitted, TAB is assumed. TAB will create a tab delimited file while CSV will create a comma delimited file. This script command is only used when writing the results from multiple ATC scenarios. The filename will be appended with the identifier for interface scenarios. If there are multiple interface scenarios, multiple files will be created.
- Case Information Displays
 - o Added new [User-Defined Case Information displays](#) which can be added to the Model Explorer
 - o Added new [Fields Pane](#) to the Model and Display Explorer giving direct access to the columns available for each case information display. You may left-click and drag fields onto the case information display to add fields.
 - o Modified Case Information Displays so that holding down the CTRL key and then dragging a column heading allows you to move column order.
 - o Added new case information displays to the [Model Explorer](#) to show Fault Analysis related information.
 - o Started work on enhancing the [plot columns](#) feature. It now allows specification of color, line type, etc... for each column.
 - o Added an explanation to the caption of the Paste button on the case information local menu if the Paste button is disabled. [The button may say "key fields missing" or "object type invalid" if the first two lines of the clipboard are incorrect](#).
- Contingency Analysis Enhancements
 - o Added a [post-contingency auxiliary file specific for particular contingency](#). If this auxiliary is specified then the [contingency option post-contingency auxiliary file](#) will not be loaded.
 - o Previously the [auto-insertion of contingencies](#) routine did not allow the auto-insertion of Generator or Bus outages at the [slack bus](#). Simulator actually handles this outage fine, so the restriction of the auto-insertion has been removed.

- o Added the ability to define [Monitoring Exceptions](#), special inclusion and exclusion monitoring lists of elements with each contingency. This allows you to NOT monitor specific elements, or to monitor only a specific list of elements.
- o Optimized the calculations done for [Linearized Contingency Analysis](#) to speed them up substantially. Linearized contingency analysis should now run approximately 3 times faster.
- o When using make-up power in the contingency analysis tool, the generation make-up loop was using a very low tolerance (4 times the mismatch solution tolerance). This was causing solution times to be much slower than desirable. We changed the default tolerance to be 5 MW, which for typical single-line outages results in a contingency solution speed between 2 and 3 times faster. The user may also adjust this tolerance in the [Contingency Modeling Options](#).
- o Added ability to apply an [Advanced Filter](#) to filter [contingency records](#) by Branch, Generator, Load, Switched Shunt, Interface, Injection Group or DC Line. A Contingency record will meet the filter if it contains any of these objects in its element actions that meet their filter.
- o Added several messages to contingency solution to notify of generator record changes in the post-contingency solution regarding the [Advanced Modeling Options](#) of Line Drop Compensation, Maximum MW Response, AGC, and Participation Factor.
- o Added the ability to properly monitor [interfaces in both directions](#) in the contingency analysis tool.
- o Added more options for [saving a list of contingencies as an auxiliary file](#). Previously the normal Save As Aux methods were overridden to save a hard-coded list of fields as well as many other related objects. The normal Save As Aux choices now behave the same as other case information displays, while two new buttons have been added that save the more complete hard-coded set of objects and fields.
- o Added options to [write out PTI CON files](#) using bus name/kvs instead of bus numbers as well as ability to specify whether to truncate names to 8 or 12 characters.
- o Enhanced the [linearized lossless contingency solution method](#) and the DC approximation contingency solution method to handle all situations when islanding occurs as a result of the contingency.
- Data Objects
 - o Added many new fields that show a count of the number of generators, loads, branches, etc... that are contained by Areas, Zones, or Substations.
 - o Added extra columns to MW Transactions to allow the import/export bid to be specified directly there.
 - o Modified so that as three-winding transformer impedances and tap ratios remain synchronized with their two-winding transformer equivalents as values are changed.
- Dialogs
 - o Modified the [Find Dialog](#) behavior when pasting in text so that it will now search to go to the nearest match when the search parameter is pasted.
 - o Added the ability on the [Power Flow Solution: General Options](#) to change the system MVA Base. This will then update all the internal structures to store the proper values on the new base.
- [Difference Flows](#) tool
 - o When using the [Difference Case](#) in Difference Flows, support was added for showing the difference of status fields so that you can see which branches or generators have changed status between cases. Fields will show up as either OPEN/CLOSED or CLOSED/OPEN if the status changed. The status in the present case will be listed first.
 - o Added option regarding whether to use Bus Numbers or Bus Name_NomKV when saving auxiliary files from the [topological differences dialog](#).
 - o [Difference Flows](#) is now available in both Edit Mode and Run Mode.
- Equivalencing
 - o Added new [option](#) Adjust Area Unspecified Interchanges to Zero Out ACE.
- Fault Analysis

- Simulator defaults all zero-sequence parameters for transmission line to be equal to [2.5 times the positive sequence impedances](#). Simulator continued this behavior when reading in PTI Sequence files, but it appears that PTI assumes that lines are open in the zero-sequence model if no zero-sequence data is specified in the *.seq data file. When reading *.seq Simulator now gives the user the ability to assume this behavior.
- File Formats
 - PowerWorld Binary File (PWB) version 14
 - PowerWorld Auxiliary File (AUX) support
 - When choosing to Save a Case, added new filetypes for saving as [AUX using secondary keys or labels](#).
 - GE EPC file support
 - When reading GE EPC files, modified it to read multi-section line interface elements directly instead of assigning the element to the first or last section of the multi-section line.
 - When reading an entire case from a GE EPC file, if loads are encountered that have the same ID except that the alphabetical case is different, a unique ID will be assigned to the current load instead of only keeping the first load encountered in the file.
 - Added more error handling when reading EPC files to handle situations where AC branch, generator, load, or switched shunt ids were duplicated at a single bus. It appears that the EPC file permits duplicate IDs if their upper/lower case is different. That EPC format also treats the circuit IDs "1" the as different than "1 " (i.e. it treats the spaces as different IDs). Simulator now handles this but also creates new unique IDs as appropriate. Messages are written to the log.
 - After reading an EPC file, Simulator will now go through and turn off control for all transformers that are more than 1/2 step size out of their Max/Min tap range.
 - Previously when reading from an EPC file Simulator would skip all records that did not contain ALL expect fields. It has now been modified to just skip the missing fields but still properly read the file.
- [Bus View Online](#)
 - Modified to show radial connections to a bus or load above the Main Bus.
 - Modified color scheme of bus links so that they are a mixture of the background color and the bus color. The old "bus link color" is no longer used.
 - When customizing the bus view, modified to allow the user to define the field prefixes also.
- Oneline Diagrams
 - Added [Line Navigation Arrows](#) to provide an easy way to pan/zoom to the other end of a transmission line or interface object.
 - The contouring algorithm has been completely rewritten to utilize advances in computer graphics cards. Contouring will now be more than 10 times faster than previous versions.
 - [Contouring](#) is now available while in either Edit Mode or Run Mode.
 - Added a [three-winding transformer symbol](#)
 - On the [PTDF dialog](#), When clicking the button to "Visualize PTDFs", the pie chart oneline options are now changed to show PTDFs also. Previously only the flow arrows were changed.
 - When inserting a bus graphically on a oneline diagram, modifications were made to provide better feedback on the [Bus dialog](#) as to whether or not a new bus is being added to the power system model.
 - When drawing onelines with substation objects instead of bus objects as the primary object, added better support for automatically anchoring generator, load and switched shunt objects to the substation.
 - Greatly increased the speed of refreshing anchors on the oneline diagram.
 - Added Pre-Defined Selections to the [Auto-Insert Borders dialog](#) that provide quick selections for North America, USA State Borders, Canadian Provinces, or Entire World.

- o Modified the [online links](#) so that they can also open Display Auxiliary Files (AXD).
- o Added option on the Draw Tab to choose [Auto Insert > Reset Stub Locations...](#) which will completely redo the positioning of objects anchored to buses and substations in the manner done when auto-inserting lines, gens, load, and shunts.
- o Added ability on the [Edit Mode Substation Information Dialog](#) to apply the latitude/longitude of the substation online display object to the substation data record.
- o Changed the circuit breaker objects to allow them to be dynamically formatted using colors.
- o Added support for generator rotors spinning counter-clockwise by enter a negative number in [angle change per refresh](#).
- o Added the ability to [contour](#) multi-section line objects. When contouring line objects, the contour image for a multi-section line will reflect the series of AC lines it represents. When contouring bus objects, locations of intermediate buses will be contoured.
- o Changed printing so that the user can pick a particular DPI output to a printer. This allows onelines to be printed at printer resolution, not only screen resolution.
- o Added three new [circuit breaker symbols](#) for Earthing Switch, Pole Top Switch, and Auto Pole Top Switch.
- Optimal Power Flow Enhancements
 - o On the Price Details case information display, modified so that the column headings regarding contingency violations include the name of the contingency also.
 - o Added support for doing Difference Flows comparisons of the OPF Reserve results.
- Power Flow Solution
 - o Improved solution performance for the situation where line X has just been CLOSED while at the same time Line Y has just been OPENED, if Line X and Line Y are in series with one another. Previously, [angle smoothing](#) was performed on the CLOSED line X in this situation which worked very poorly. This is now handled using a more appropriate voltage and angle estimation.
 - o Added restrictions to phase-shifter transformer movement in the Control Loop of the [power flow solution](#) to restrict larger phase changes across a very small impedance phase-shifter. This helps with power flow convergence.
 - o When [dynamically assigning new slack buses](#), Simulator chooses the bus that has the largest maximum generator MW output. This has been changed so that buses with a maximum generator MW output larger than 5000 are ignored because these likely represent generators with fictitious MW limits such as 9999.
 - o Added special [Wind Control Mode](#) options for generators which effect how the Mvar limits are handled.
- PV and QV Curve Enhancements
 - o Added [Reverse Transfer Option](#).
 - o Completely rewrote the [PV stepping algorithm](#) to avoid unnecessary power flow solutions.
 - o Modified [Quantities to Track](#) to allow the inclusion of ATC Extra Monitors in the tracking.
 - o Removed the "Study Name" from the Auxiliary format [SCRIPT](#) and DATA sections. This is not needed anymore.
 - o Added support for Custom Expressions in the QV results case information display.
 - o Added the ability to [specify merit order dispatch separately](#) for the source and sink. This now allows you to use merit order for one and NOT the other.
 - o When ramping in the PV curve tool, added the ability to [specify ZIP components](#) for Q (MVAR) load.
 - o When ramping in the PV curve tool, added the ability to [scale load in proportion to existing ZIP ratios](#).
 - o Added support for advanced filtering on the PV plot form when selecting a list of objects.
- Sensitivity Calculations
 - o Added a list of Injection Groups to the [TLR/GSF dialog](#).
 - o Added support for extracting the [multiple element TLR](#) sensitivity columns through SimAuto and through script commands

- o Added a new tab for showing [multi-element injection group sensitivities](#) on the TLR/GSF dialog. The values shown will be calculated from the bus values with weightings determined by the injection group participation factors.
- o Added a new tab for showing Load sensitivities on the [TLR/GSF dialogs](#). Sensitivities displayed represent the sensitivity of the loads terminal bus.
- Time Step Simulation Enhancements
 - o Added Auxiliary File Support for [Schedule Subscriptions](#).
 - o Added support for Time Step Simulation [schedule subscriptions](#) from DC Line fields.
 - o Added support for [saving schedule subscriptions](#) to aux files, including fields from generators bid points.
 - o Modified the TSB file format to save the start and end time points of a simulation.

Help

On-line help is available in PowerWorld Simulator via the **Help** ribbon group on the [Window](#) ribbon tab, or by pressing the **F1** key on many dialogs and displays. Context-sensitive help is available on the oneline diagrams. To obtain object specific help, position the mouse over the object in question on the oneline and press the **F1** key.

Sample power flow cases and other information are available at the PowerWorld web site:
<http://www.powerworld.com>

Contact technical support at support@powerworld.com for answers to your questions regarding any PowerWorld product. Or call us at (217) 384-6330.

PowerWorld Corporation

2001 S. First Street
Suite 203
Champaign, IL 61820

Contact Information

PowerWorld Corporation
2001 South First Street
Champaign, IL 61820

Phone: (217) 384-6330
Fax: (217) 384-6329
Toll Free: (877) 748-7840 - U.S. and Canada only

General Information: info@powerworld.com
Technical Support: support@powerworld.com
Sales: sales@powerworld.com

www.powerworld.com

Windows Basics

Operating System

Simulator runs under all Windows 2000 and later Microsoft Windows Operating Systems.

Mouse Conventions

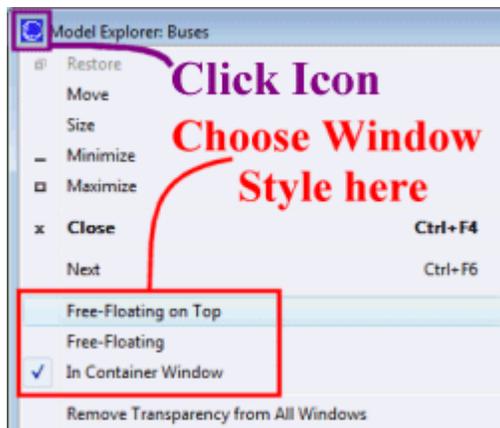
Since much of the interaction between Simulator and the user is accomplished by using the mouse, we have designed the interface to obey consistent conventions for mouse usage. In general, the left-mouse button is used to affect some sort of immediate change or control over a power system element, while the right mouse button is used to gain more information about a power system element or to view a list of available options. More details on are provided throughout this manual.

Dialog Window Position, Size Tracking

As various dialogs are opened, PowerWorld will keep track of the Free-Floating/Contained style as well as the position and size of each dialog. When that dialog is then opened again it will open to the same style, size, and position as it was previously. These settings are stored in the Windows registry so that they are remembered when the program is opened again. You can always reset all these settings to their factory defaults by clicking the **Reset to Default** button on the Window Ribbon Group in the [Windows Ribbon Tab](#).

Dialog Window Styles (Free-Floating, Free-Floating On Top, and In Container)

By default, most dialogs or windows when opened are contained inside the main program window. (The only general exception to this are dialogs that are opened as "modal" windows which you must close before continuing to use the software.) You can change this default globally by choosing **Switch to Free-Floating Windows** on the Window Ribbon Group in the [Windows Ribbon Tab](#). Alternatively, you can leave most windows in the container and individually choose to make specific windows Free-Floating. This is done by left clicking on the icon in the upper left of the dialog to open the windows system menu. This is allowed for all windows except for oneline diagrams. (Online diagrams always appear inside the main container window.) For windows that we allow this for, three choices will appear as shown below.



In Container Window

In Container Window is the same as the default behavior for most windows. The dialog will be contained inside the main program shell and can not be moved outside of it.

Free-Floating on Top

Setting a window or dialog to Free-Floating on Top will make the window float independent of the main program shell. The dialog will also always remain on top of the main program shell and thus can not get lost behind the main program shell. The [message log](#) window is one of a small number of windows that defaults to this behavior.

Free Floating

This style is the same as Free-Floating on Top, however the dialog will become hidden behind the main program window when the main program window becomes active.

Example Windows Styles

Free-Floating

The screenshot displays the PowerWorld software interface with several windows and annotations:

- Area/Zone/Owner Filters**: A floating window at the top with a red border, containing the text "Set all Area/Zone/Owners to 'Yes'" and a "Close" button. A red arrow points to it from the "Free-Floating" label.
- Main Application Window**: The central window titled "Case: Sample.pwb Status: Initialized | Simulator 14 Beta". It features a menu bar (Case Information, Draw, Onelines, Tools, Options, Add Ons, Window) and a toolbar with buttons like "Run Mode", "Model Explorer...", "Area/Zone Filters...", and "Limit Monitoring...".
- Model Explorer: Buses**: A window with a purple border showing a tree view on the left and a "Bus Records" table on the right. A purple arrow points to it from the "Free-Floating on Top" label.
- Case Hourly Cost**: A window with a green border showing a network diagram and cost information. A green arrow points to it from the "In Container Window" label.

Number	Name	Area Name	Nom kV	PU Volt
1	One	Top	138.00	1.050
2	Two	Top	138.00	1.040
3	Three	Top	138.00	0.992
4	Four	Top	138.00	1.000
5	Five	Top	138.00	1.006
6	Six	Left	138.00	1.040
7	Seven	Right	138.00	1.040

Case Hourly Cost: 16939 \$/h

Network diagram details: 80 MW, 30 Mvar, 32 MVV, 14 MVV, 95 MW AGC ON, 130 MW, 40 Mvar, 1.00 pu, 1.01 pu, Cavan.

Getting Started With PowerWorld

The screenshot displays the PowerWorld software interface. At the top, the title bar reads "Case: Sample.pwb Status: Initialized | Simulator 14 Beta". Below the title bar is a menu bar with options: Case Information, Draw, Onelines, Tools, Options, Add Ons, and Window. A toolbar contains various icons and buttons, including "Run Mode".

Several windows are open and stacked on top of each other:

- Sample**: A window showing a power system diagram with a 102 MW generator and a bus labeled AGC ON.
- Area/Zone/Owner Filters**: A dialog box with a button "Set all Area/Zone/Owners to 'Yes'" and a "Close" button.
- Bus Records**: A data table window showing bus information.
- Area Name Summary**: A small table showing the number of buses for each area name.

The **Bus Records** window contains the following data:

Number	Name	Area Name	Nom kV	PU Volt	Volt (kV)
1	One	Top	138.00	1.05000	144.9
2	Two	Top	138.00	1.04000	143.5
3	Three	Top	138.00	0.99269	136.9
4	Four	Top	138.00	1.00000	138.0
5	Five	Top	138.00	1.00665	138.9
6	Six	Left	138.00	1.04000	143.5
7	Seven	Right	138.00		

The **Area Name Summary** window shows:

Area Name	Shown	# of Buses	Min Bus N
	YES	5	
	YES	1	
	YES	1	

Overlaid on the bottom right of the screenshot is the text: **All Windows are In Container Window**.

PowerWorld Simulator: Getting Started

The key to using Simulator is to recognize that it has two distinct modes **Edit Mode** and **Run Mode**. The Edit Mode is used to construct new simulation cases or to modify existing cases, while the Run Mode is used to perform the actual power system simulation. You can easily switch between the modes using the **Edit Mode** and **Run Mode** buttons in the **Mode** ribbon group.

If you are new to Simulator and seek a quick means of familiarizing yourself with it, we recommend starting with the tutorials; see [Creating a New Case](#) or [Starting with an Existing Case](#)

Sample cases are provided with the software in the "Sample Cases" directory. If you're interested in learning by doing, you may wish to just [open one of the sample cases](#) and try the different Simulator features.

Edit Mode Introduction

The Edit Mode is used to create a new case or to modify existing cases. To switch to Edit Mode, click on the **Edit Mode** button in the **Mode** ribbon group on any of the ribbon tabs.

Here is a sampling of things you can do in Edit Mode:

- Create a new case; see [New Case](#) for details.
- Create a new oneline diagram; see [New Oneline](#) for details.
- Add new components graphically to an existing case; see [Draw](#) ribbon tab.
- Modify the appearance of the oneline objects; see [Formatting](#) ribbon group for details.
- View and modify a case using non-graphical lists displays; see [Case Information Displays](#) for details.
- Equivalence a case; see [Equivalencing](#) for details.
- Append a subsystem to an existing case; see [Appending a Case](#) for details.

For more details on the Edit Mode please see [Edit Mode Overview](#).

Run Mode Introduction

The Run Mode is used to solve a single Power Flow Solution, run one of the available load flow tools, or run a time-domain simulation of the power system. To access the Run Mode, click on the **Run Mode** button in the **Mode** group of the Simulator Ribbon.

The key menu associated with the Run Mode is the [Simulation](#) menu. This menu allows you to perform a single Power Flow Solution (however, it is quicker to use the [Quick Access](#) toolbar).

Other key components of the Run Mode include:

- The oneline diagrams, which allow you to view the case graphically. See [Online Diagram Overview](#) for details.
- The Case Information Displays, which allow you to view the entire power system case using list displays. See [Case Information Displays](#) for details.
- Dialogs to change the simulation options and the Power Flow Solution. See [Simulation Options](#) for details.
- Scaling to allow easy variation in the load, shunts, and generation at any number of buses. See [Scaling](#) for details.
- Contouring, which shows a color contour representing the variation in any power system parameter across a system. See [Contouring](#) for details.
- Transfer distribution factor calculations. See [Power Transfer Distribution Factors](#) for details.
- Perform a fault analysis. See [Fault Analysis](#) for details.
- Run [Transfer Capability](#) studies.
- Perform an [Optimal Power Flow](#) (OPF) or [Security Constrained Optimal Power Flow](#) (SCOPF) analysis.
- Generate [PV and QV curves](#).

Script Mode Introduction

The Script Mode is used to access the Script Command window. Simulator scripting allows a method of grouping multiple commands for sequential processing by Simulator. From the [Script Command Execution](#) window, the user can manually enter [script commands](#) for processing, or load an [auxiliary file](#) containing multiple script commands and data modification commands.

Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language, but with the addition of a **submode** feature. Submodes determine which script commands can be called. Only those commands available to the current submode can be executed. You will always be in one of the submodes when executing a script. If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to the **EDIT, CASE** submode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to the **RUN, POWERFLOW** submode.

To switch submodes, use the [EnterMode \(mode or submode\)](#) script command. The following list includes the submodes available to each of the Simulator modes. Click on the links below to see the actions available to each mode and/or submode.

[Edit Mode](#)

- [Case Submode](#)

[Run Mode](#)

- [PowerFlow Submode](#)
- [Contingency Submode](#)
- [ATC Submode](#)
- [Fault Submode](#)
- [PV Submode](#)
- [QV Submode](#)

There are also a number of general script actions that can be executed in any of the submodes. See [Script General Actions](#) for more details.

Message Log

The Message Log displays detailed results of each Power Flow Solution, chronicling the solution process iteration by iteration. It also reports messages raised by Simulator in performing various operations, such as opening or validating a case. The Message Log can be helpful when you run into problems solving a particular simulation case. The Message Log is not used with Viewer.

To display the Message Log, go to the [Tools](#) ribbon tab, and click the button  **Log** on the **Log** ribbon group. This button is also available on the [Add Ons](#) ribbon tab.

The Message Log is one the few windows that is set by default to be **Free-Floating On Top**. This means that the dialog floats independently of the main program window and always remains on top of the main container window. If you would like to change this style to make the message log contained inside the container window, see the [Window Basics](#) help for more information.

Message Log Local Menu

Right-clicking on the Message Log displays its local menu. To change the font characteristics of the log, select **Change Font**.

To suppress or change color of [certain messages associated with the power flow case](#) select either **Suppress Messages** or **Color Messages** and then the desired message whose characteristics want to change. For more information about the colors and messages see the [Message Log Options](#).

You can find certain text by clicking on **Find**, and then entering the text in the dialog displayed. Also, you can highlight all the text in the log, or inversely unselect all the highlighted text by clicking on **Select All**, or **Unselect All**, respectively. To print or copy the contents to the Windows clipboard of a highlighted section of the message log, select **Print Selection** or **Copy Selection to Clipboard**. You can also clear the contents of the log by selecting **Clear Window**. Lastly, you will find some log options available, such as **Change Maximum Lines**, **Disable Logging**, **Disable Time Stamp**, and **Save Log to File** options which allow automatically saving the log to a file during load flow analysis.

Message Log Color Outlines and Color Background

When solving the power flow, there are several loops which occur that are discussed the [Solving the Power Flow](#) Topic. These loops will generate **Red**, **Purple**, **Green**, and **Blue** outlines on the message log to visually indicate which loop the message text is generated by. This helps in determining which part of a the power flow solution may be having difficulty solving.

Additionally, when the various tools and add-ons send message text to the log the background of the text will be colored to indicate this. For instance, contingency analysis actions will be highlighted with a **light blue** background, while PV, QV, ATC, and OPF add-on message text will be highlighted with a **light yellow** background.

Custom and Memo Display

The Custom tab is now available on many dialogs and displays in Simulator. The purpose of the custom tab is to allow the user to add their own custom information and comments to data and objects in the load flow case. Simply switch to the Memo or Custom page on a display and start typing in the memo box, or add values or text in the custom fields. The information added to these memo pages is stored with the objects in the load flow case and saved to [PWB](#) files.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the object. Defining custom fields is detailed in [Custom Field Descriptions](#).

The custom section also contains a **Selected?** check box. This is a special field that is associated with most objects in Simulator. In many of the places where filtering is allowed, filtering can be done based on the **Selected?** field. Additional information about this field can be found under the [Set Selected Field](#) topic.

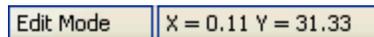
The memo section of the dialog is simply a location to log information about the object. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Status Bar

The Status Bar is displayed across the bottom of the PowerWorld Simulator window. The left-most field of the status bar displays the current Simulator mode (Edit or Run). The remaining fields vary depending on which mode of operation Simulator is currently in. The status bar displays Tool Tips when the cursor is positioned over a toolbar button or menu item. The tool tip is also shown next to the cursor after a short time delay.

Edit Mode

The Edit Mode status bar displays the [Screen Coordinates](#) of the cursor when positioned over a oneline diagram.



Edit Mode Status Bar

Run Mode

The left side of the Run Mode status bar displays simulation status ("Solution Animation Stopped" or "Solution Animation Running"), AC or DC depending on solution options and the [Difference Flows](#) status ("Current Case", "Base Case", or "Difference Case").



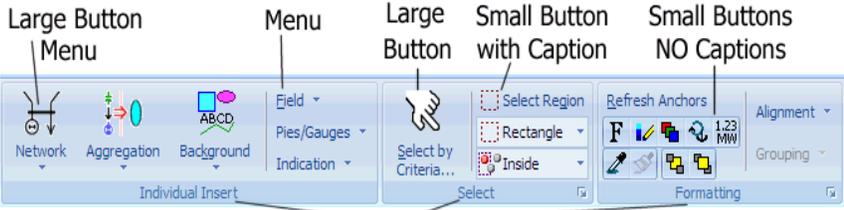
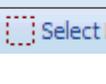
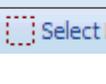
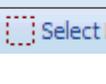
Run Mode Status Bar (Left Side)

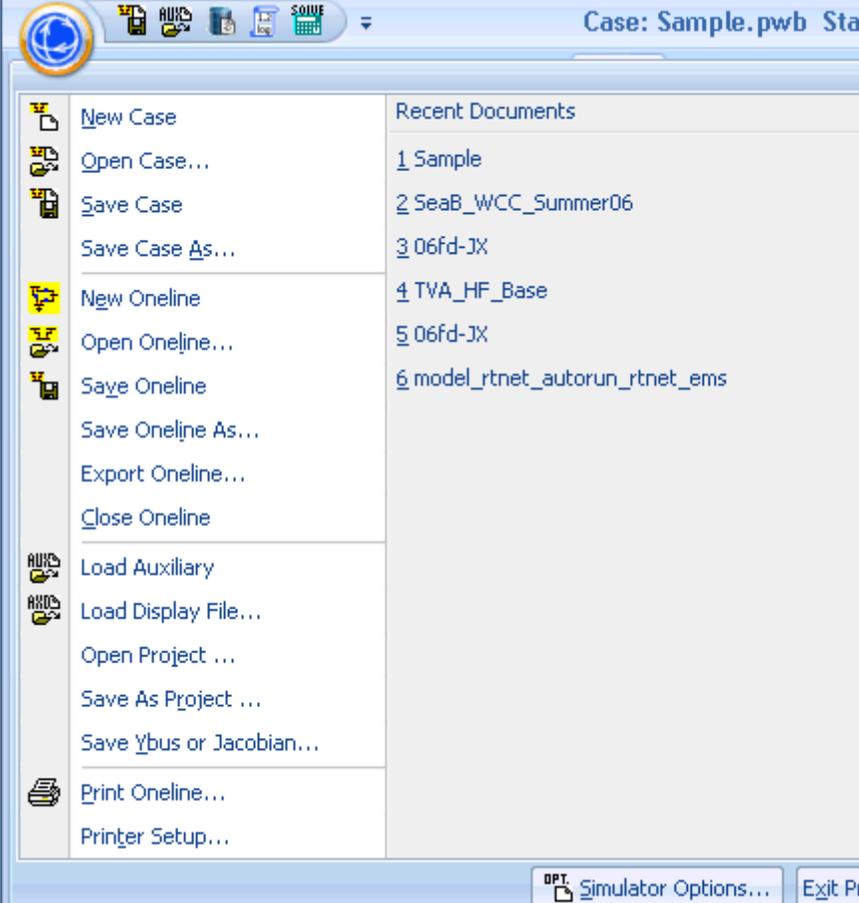
Ribbons User Interface

PowerWorld's user interface consists of a ribbon across the top of the program. Important terminology regarding the ribbon is depicted in the images below.



Ribbon	The entire strip across the top is called the ribbon
Ribbon Tab	<p>The ribbon consists of several ribbon tabs which group together features of the software that are used together. There are seven ribbons which have their own purposes and own help topics listed as follows.</p> <ul style="list-style-type: none"> • Case Information Ribbon Tab: primarily used to navigate and look through all the data in your model. • Draw Ribbon Tab: primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in Edit Mode. • Onelines Ribbon Tab: primarily used after you have already created a oneline diagram. This ribbon provides features for customizing the appearance of your oneline diagram. • Options Ribbon Tab: all of the buttons on this ribbon are also available on one of the other ribbons, however this ribbon brings all the options in the software into one place. • Tools Ribbon Tab: this ribbon provides access to all of the analysis tools that are available in the base package of PowerWorld Simulator. You will use this ribbon when you are performing power flow analysis, contingency analysis, or using the sensitivities tools for instance. • Add Ons Ribbon Tab: this ribbon provides access to all the add-on tools available for Simulator including the OPF, SCOPF, ATC, and PVQV tools. If you have not purchased these tools then the options will be grayed out. • Windows Ribbon Tab: this ribbon provides access to customizing the Windows in the User Interface. It also has some information reading help topics . <p>The goal in designing these ribbons was to create several task specific ribbons which represent the various ways that the software is used. Our hope is that you will not spend a lot of time flipping back and forth between ribbons. If you find yourself doing this please contact</p>

	<p>PowerWorld Corporation to help us make the software better for the future.</p>												
<p>Ribbon Group</p>	<p>Inside a particular ribbon tab, there are around 4 to 7 groupings called ribbon groups. The following image shows three ribbon groups (Individual Insert, Select, and Formatting) with various buttons and menus available.</p>  <p style="text-align: center;">3 Ribbon Groups</p> <p>ribbon groups join sets of buttons and menus that are related to one another. The large buttons and icons are used to draw attention to the most commonly used features of the software. All features are still available through drop-down menus in the same manner as was available previously. Our design goal in creating and laying out the ribbons was to provide the new user with insight about using the software, while at the same time giving the experienced user quick access to common features.</p> <table border="1" data-bbox="738 976 1112 1617"> <tr> <td>Large Button</td> <td></td> </tr> <tr> <td>Large Button Menu</td> <td></td> </tr> <tr> <td>Small Button With Caption</td> <td></td> </tr> <tr> <td>Small Button Menu</td> <td></td> </tr> <tr> <td>Small Button without Caption</td> <td></td> </tr> <tr> <td>Menu Only Caption</td> <td></td> </tr> </table>	Large Button		Large Button Menu		Small Button With Caption		Small Button Menu		Small Button without Caption		Menu Only Caption	
Large Button													
Large Button Menu													
Small Button With Caption													
Small Button Menu													
Small Button without Caption													
Menu Only Caption													
<p>Help Button</p> 	<p>Clicking the help button on the right edge of the ribbon brings up this online help</p>												

<p>Quick Access Toolbar</p>	<p>The quick access toolbar at the top left of the ribbon (near the Application Button) contains small buttons with no caption that are always visible. For more information about how to use and customize this see the Quick Access Toolbar help.</p> 
<p>Application Title Bar</p>	<p>Case: Sample.pwb Status: Initialized Simulator 13 SCOPF, ATC, PVQV,</p>
<p>Application Button</p> 	<p>Click on this to bring up the Application File Menu shown below. For more information about using the Application File Menu see the Application File Menu help.</p> 

Quick Access Toolbar

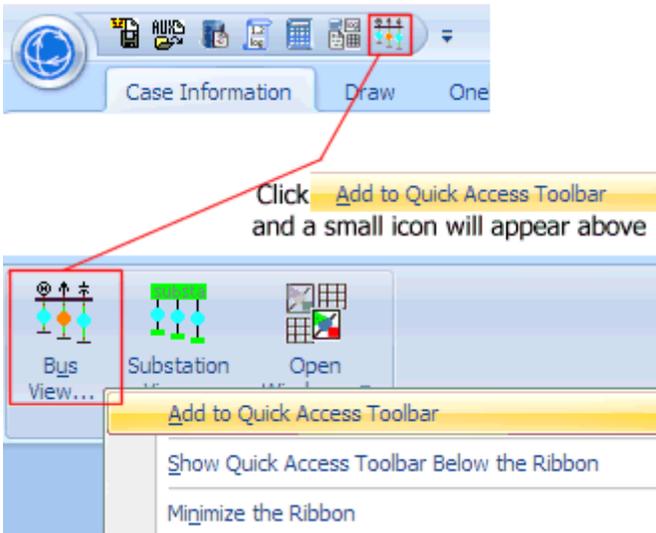
The Ribbon Quick Access Toolbar provides you with quick access to the most frequently used buttons. These buttons are found in the top left portion of the Ribbon just to the right of the [Application Button](#). The default buttons that appear in the Quick Access Toolbar are shown in the figure below.



The eight default buttons are as follows from left to right.

- [Save Case](#)
- [Open Auxiliary File](#)
- [Model Explorer](#)
- [Display Bus View](#)
- [Message Log](#)
- [Simulator Options](#)
- [Single Solution](#)
- [Abort](#)

The Quick Access Toolbar can also be customized very simply. If you find a button that you would like quicker access to, simply right-click on the button and then choose Add to Quick Access Toolbar. The small icon version of the button will then immediately appear up in the Quick Access Toolbar. This is depicted in the image below.



The Quick Access Toolbar can also be modified to show the Toolbar Below the Ribbon providing more space for specifying buttons.

Also experiment with Showing the Quick Access Toolbar below the Ribbon or Minimizing the Ribbon.

Add Ons Ribbon Tab

The Add Ons ribbon consists of buttons and menus that access to the various Simulator Add On Tools. The ribbon is shown below when configured for the OPF, SCOPF, PVQV, ATC, Transient Stability, and Topology Processing add on tools.



The Tools ribbon is broken into seven ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Log Ribbon Group

Abort

Terminates the current Power Flow Solution. If the application is performing a timed simulation, pressing the abort button will pause the simulation. See [PowerWorld Simulation Control](#) for more details.

Log

Toggles the display of the [message log](#) window. The log window shows what is going on with the Power Flow Solution process and may prove useful when you are trying to track down a problem with a non-converging model.

Script

Opens the Script dialog, which can be used to call [script commands](#) or open [auxiliary files](#) containing script commands and data modifications. Note that the drop-down next to the Script button give convenient access to the [Quick Auxiliary Files](#).

Optimal Power Flow (OPF) Ribbon Group

Primal LP

Click this button to solve the OPF. A more detailed description of what this does is found in the topic [OPF LP Primal](#).

Security Constrained OPF

Click this button to open the [Security Constrained OPF Dialog](#).

OPF Options and Results

Click this button to open the [OPF Options and Results Dialog](#).

OPF Case Info Menu

The OPF Case Info Menu provides direct access to several case information displays that show default columns related to input and output of the Optimal Power Flow. Instead of using this menu, PowerWorld encourages you leave the [Model Explorer](#) open and navigate using the Explore Pane on the Model Explorer. All of these options are available under the Optimal Power Flow folder on the Model Explorer.



PV and QV Curve (PVQV) Ribbon Group

PV Curves

Click this button to open the [PV Curves dialog](#).

QV Curves

Click this button to open the [QV Curves dialog](#).

Refine Model

Click this button to open the [Refine Model Dialog](#).

Transient Stability (TS) Ribbon Group

Transient Stability

Click this button to open the [Transient Stability Analysis dialog](#).

Stability Case Info Menu

The [Stability Case Info Menu](#) provides direct access to several case information displays that show default columns related to input and output of the transient stability tool.

Topology Processing Ribbon Group

Topology Processing

Click this button to open the [Topology Processing Dialog](#).

Available Transfer Capability Ribbon Group

Available Transfer Capability

Click this button to open the [Available Transfer Capability Dialog](#).

Case Information Ribbon Tab

The Case Information ribbon consists of buttons and menus that provide access to the information about the power system model. The ribbon is shown below



The Case Information ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Case Information Ribbon Group

Model Explorer



Opens the [Model Explorer](#) giving access to all the [case information displays](#) for the model.

Area/Zone Filters



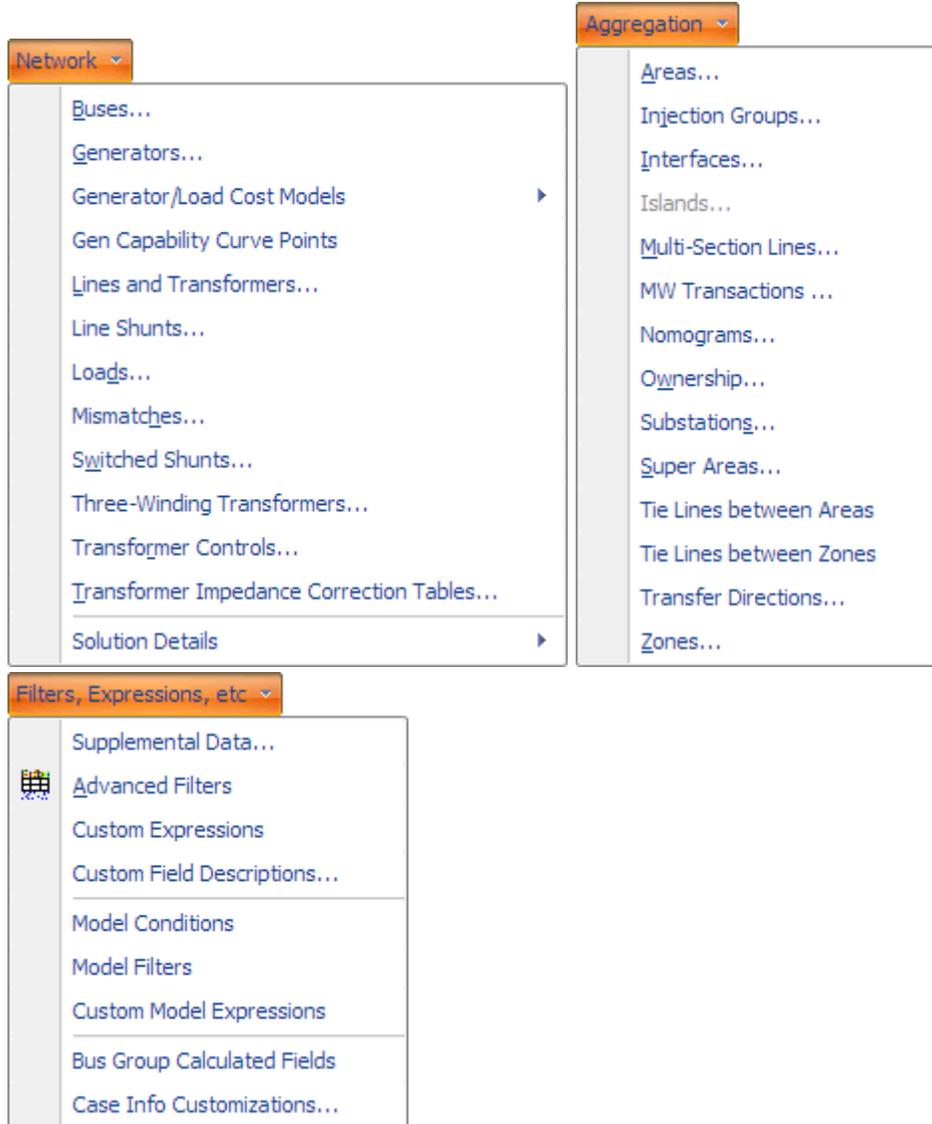
Opens the [Area/Zone/Owner Filters](#) display which allows filtering of the case information displays by Area, Zone, and Owner.

Limit Monitoring

Opens the [Limit Monitoring Settings and Limit Violations Dialog](#) which allows you to specify what to monitor in the various analysis tools.

Menus for Network, Aggregation, and Filters, Expression, Etc

These three menus all perform the same action as opening the Model Explorer, except that they automatically set the Explore Pane of the Model Explorer to show the respective case information display. Instead of using these menus, PowerWorld encourages you leave the Model Explorer open and navigate using the Explore Pane on the Model Explorer.



Case Data Ribbon Group

Difference Flows Menu



Difference Flows: Opens the [Difference Flows Dialog](#)

Present Topological Differences From Base Case: Opens the [Present Topological Differences From Base Case Dialog](#).

The remaining options in this menu all replicate the same behavior available on the [Difference Flows Dialog](#). For more details so that dialog.

Simulator Options

Opens the [Simulator Options Dialog](#).

Case Description

Opens the [Case Description Dialog](#).

Case Summary

Opens the [Case Summary Dialog](#).

Custom Case Info

Opens the [Custom Case Information Display](#)

Power Flow List

Opens the [Power Flow List](#)

Quick Power Flow List

Opens the [Quick Power Flow List](#)

AUX Export Format Desc

Opens the [Auxiliary File Export Format Description Dialog](#) for Power System Objects.

Views Ribbon Group

Bus View

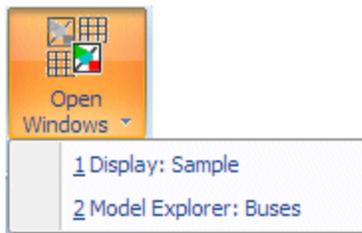
Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Substation View

Opens the [Substation View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that substation.

Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



Draw Ribbon Tab

The Draw ribbon tab is primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in [Edit Mode](#). The Draw ribbon tab is shown below.



The Draw ribbon consists of seven ribbon groups described as follows

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. For detailed help see the [Quick Insert](#) ribbon group. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic.

Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert](#) ribbon group. However, for complete access to all objects including those that can not be "quick - inserted", see the help under the [Individual Insert](#) ribbon group topic.

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. For more detailed help see the [Selection](#) ribbon group topic for more detailed help.

Formatting Ribbon Group

The Format ribbon group contains options for formatting the selected objects on your active oneline. For more detailed help see the [Formatting](#) ribbon group topic for more detailed help.

Clipboard Ribbon Group

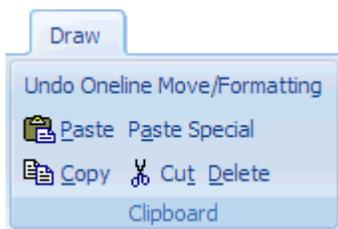
The Clipboard ribbon group contains options for copying and pasting oneline display objects. For more detailed help see the [Clipboard](#) ribbon group topic for more detailed help.

Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline objects while your are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



Undo

The Undo command is used in the edit mode to undo the last change made on the oneline diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the oneline. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

Absolute Coordinates : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different oneline diagrams which contain the same geographic background.

Coordinates Relative to Cursor : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

Formatting Ribbon Group

The **Formatting** ribbon group contains options for formatting the selected objects on your active oneline. This group is on the [Draw](#) ribbon tab and its options are available only in Edit mode. The options allow you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. The group is shown below.



A description of the actions on this group are as follows.



Format Buttons

Clicking this button will open the [Format Dialog](#). The dialog which opens will have five tabs as follows.

Font

Select this option to change the font with which the selected objects are displayed. The [Font Tab](#) of the Format Selection Dialog is displayed.

Line/Fill

Select this option to change the line thickness and style with which a display object is drawn, or the fill color for closed shapes. The [Line/Fill Tab](#) of the Format Selection Dialog is displayed.

Levels/Layers

Select this option to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize. The [Levels/Layers Tab](#) of the Format Selection Dialog is displayed.

Display/Size

Simulator can change the attributes of the way multiple objects are displayed using this option, which opens the [Display/Size page](#) of the Format Selection Dialog. The size of the objects can be adjusted, as well as the orientation of the object(s) from their terminal buses. Objects can also be Anchored to their terminal buses or devices, or marked as Immobile, meaning the object(s) cannot be moved on the oneline diagram.

Field

Select this option to change the attributes related to object text fields which are selected. The [Field Tab](#) of the Format Selection Dialog is displayed.

Anchors

Invokes a command to [refresh anchors](#) on all oneline objects.



Copy Format and Paste Format Buttons

The button on the left is the **Copy Format** button. After selecting an object this will copy all the format properties of that object into an internal buffer. When you then subsequently select other objects, you can click on the button on the right which is the **Paste Format** button. Clicking the **Paste Format** button will bring up the [Paste Format Dialog](#).



Send to Back, Bring to Front Buttons

After selecting a group of oneline objects using any of the methods described on the [Select](#) ribbon group, you may then click the **Send to Back** button to cause all the objects selected to move to underneath other objects. Click the **Bring to Front** button to bring objects to the top.

The Send to Back and Bring to Front menu options govern the visibility of display objects that occupy the same screen stack level. All objects have an associated screen stack level, which may be one of Base, Background, Middle, or Top. Objects having a screen stack level of Top obscure all objects having stack levels of Middle, Background, or Base that occupy the same region of the screen. They may or may not obscure objects having the same stack level of Top depending on the order in which the objects having the same stack level are drawn. Selecting Send to Back for a selected object will cause it to be obscured by all other same-level objects that occupy its location. Selecting Bring to Front for a selected object will cause it to be drawn above all other same-level objects that occupy the same region of the screen. The Send to Back and Bring to Front menu options govern relative placement of objects only *within* stack levels. The Send to Back and Bring to Front options do not affect the relative placement of objects having different stack levels.

Alignment Menu

The alignment menu provides access to the [Alignment Dialog](#) and direct access to options to align the selected objects as specified in the menu items.

Grouping Menu

Grouping is only available for background oneline objects (background lines, ellipses, text, etc...). This menu provides button to Group and Un-Group objects.

Layers Menu

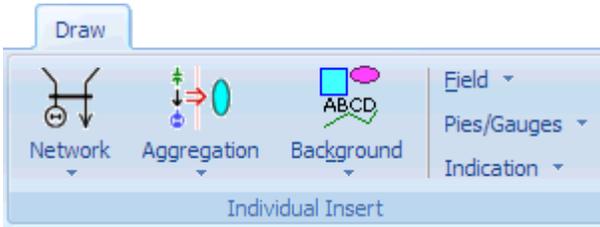
The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert Group](#). The Individual Insert group is shown below.



The six menus available on the group categorize the objects that can be placed on the oneline diagrams. A summary of what is in each menu is described below along with a picture of each of the six menus.

Network Menu

The objects in this menu represent network model objects. Generally these represent actual physical devices such as a generator or transmission line. They are similar to the model objects found in the **Network** Folder on the Explore Pane of the [Model Explorer](#).

Aggregation Menu

The objects in this menu represent model objects that represent an aggregation of other model objects such as interfaces or injection groups. They are similar to the data objects found in the **Aggregation** Folder on the Explore Pane of the [Model Explorer](#).

Background Menu

The objects in this menu represent objects that are only background text and do not link to any of the objects you would look at in the [Model Explorer](#). Some of the objects link to external objects such as documents, but not to elements of the model.

Pies/Gauges Menu

The objects in this menu represent pie charts or gauges which link to the model data found in the [Model Explorer](#)

Field Menu

The objects in this menu represent fields which link to the model data found in the [Model Explorer](#)

Indication Menu

The objects in this menu represent indications of status or flow for objects that are found in the model data found in the [Model Explorer](#)

Network

- [Bus](#)
- [Generator](#)
- [Load](#)
- [Switched Shunt](#)
- [Transmission Line](#)
- [Transformer](#)
- [Series Capacitor](#)
- [DC Transmission Line](#)

Aggregation

- [Area](#)
- [Injection Group](#)
- [Interface](#)
- [MS Transmission Line](#)
- [Owner](#)
- [Substation](#)
- [Super Area](#)
- [Zone](#)

Background

- [Text](#)
- [Memo Text](#)
- [Online Link](#)
- [Document Link](#)
- [Picture](#)
- [Background Line](#)
- [Background Rectangle](#)
- [Background Ellipse](#)
- [Insert GIS Data From Shapefile](#)

Pies/Gauges

- [Line Flow Pie Chart](#)
- [Interface Pie Chart](#)
- [MS Line Flow Pie Chart](#)
-
- [Bus Pie Chart/Gauge](#)
- [Generator Pie Chart/Gauge](#)
- [Switched Shunt Pie Chart/Gauge](#)
- [Transformer Pie Chart/Gauge](#)
-
- [Area Pie Chart/Gauge](#)
- [Injection Group Pie Chart/Gauge](#)
- [Owner Pie Chart/Gauge](#)
- [Substation Pie Chart/Gauge](#)
- [Super Area Pie Chart/Gauge](#)
- [Zone Pie Chart/Gauge](#)
-
- [Old Gauges](#)

Field

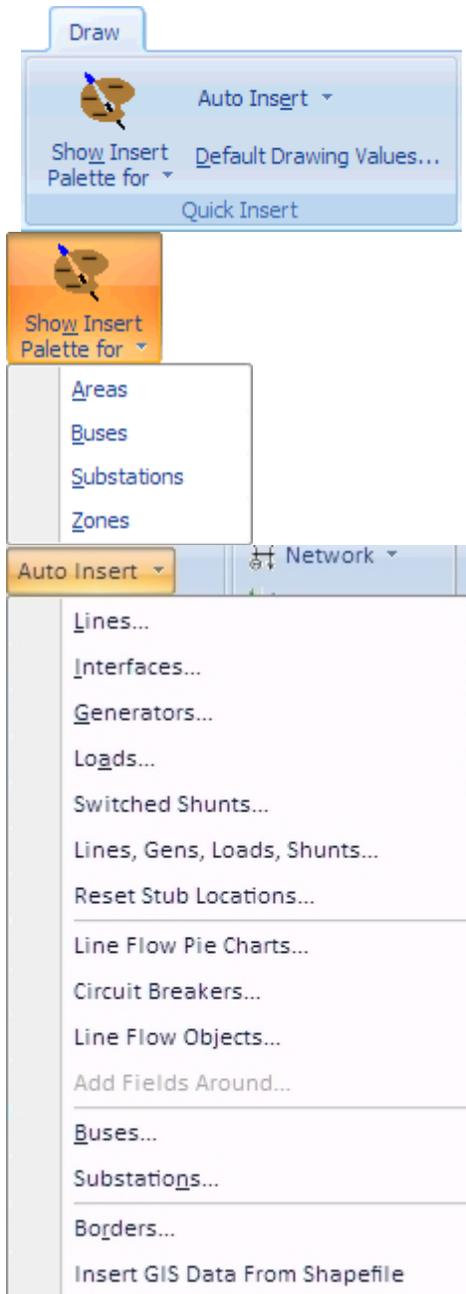
- [Bus Field](#)
- [Generator Field](#)
- [Load Field](#)
- [Switched Shunt Field](#)
- [Transmission Line Field](#)
- [DC Transmission Line Field](#)
- [MS Transmission Line Field](#)
- [Series Capacitor Field](#)
- [Transformer Field](#)
-
- [Area Field](#)
- [Interface Field](#)
- [Substation Field](#)
- [Owner Field](#)
- [Zone Field](#)
- [Super Area Field](#)
-
- [Case Information Memo Field](#)
- [Generic Model Field](#)
- [Online Field](#)
- [Supplemental Data Field](#)

Indication

- [Circuit Breaker](#)
- [Line Flow Arrow](#)

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating an online diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. If you would instead like help on inserting a single object on your online diagram see the [Individual Insert](#) ribbon group topic. Both of the ribbon groups are on the [Draw](#) ribbon tab. The Quick Insert ribbon group is shown as follows.



Show Insert Palette for Menu

Choosing any of the items in the Show Insert Palette for Menu will open up its respective Insert Palette. This Insert palettes show a list of all objects that are already shown on the online diagram as well as a list of objects which are not. For more information see the [Using the Insert Palettes](#) help topic.

Show Insert Palette for Menu

The first six options are used to automatically insert network display objects that are connected to bus objects that have already been drawn on the online.

Lines : Shows the [Automatically Insert Lines Dialog](#)

Interfaces : Shows the [Automatically Insert Interfaces Dialog](#)

Generators : Shows the [Automatically Insert Generators Dialog](#)

Loads : Shows the [Automatically Insert Loads Dialog](#)

Switched Shunts : Shows the [Automatically Insert Switched Shunts Dialog](#)

Lines, Gens, Loads, Shunts : Shows a dialog for auto-inserting lines, generators, loads, and shunts simultaneously.

Reset Stub Locations : Will completely redo the positioning of objects anchored to buses and substations in the manner done when auto-inserting lines, gens, load, and switched shunts.

The next four options provide methods for adding pie charts, fields, circuit breakers and line flow objects

around objects that have already been drawn on the oneline diagram.

Line Flow Pie Charts : Select this to automatically insert [line flow pie charts](#) on the transmission lines or transformers objects that are selected.

Circuit Breakers : Select this to automatically insert [circuit breakers](#) on the transmission lines or transformers objects that are selected.

Line Flow Objects : Select this option to automatically insert line flow objects on the transmission lines or transformers objects that are selected.

Add Fields Around : Select this option to insert fields around the presently selected objects. It will then bring up the [Inserting and Placing Multiple Display Objects Dialog](#)

The follow two options provide a mechanism for auto-inserting bus or substation oneline objects. You must have geographic longitude/latitude information in order to use these options.

Buses : Shows the [Automatically Insert Buses Dialog](#)

Substations : Shows the [Automatically Insert Substations Dialog](#)

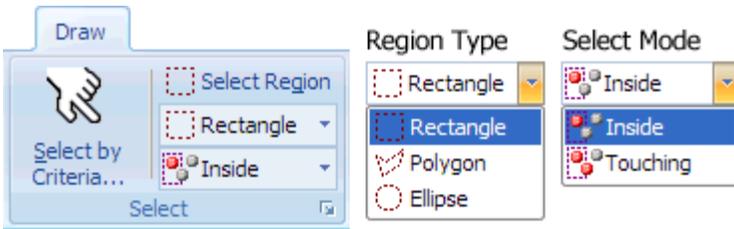
The last two options provide a mechanism for inserting background lines that represent geographic borders or from a Shapefile.

Borders : Shows the [Automatically Insert Borders Dialog](#)

Insert GIS Data From Shapefile : Shows the [Shapefile Import Dialog](#)

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. The features in this topic are only available when you are in [Edit Mode](#). This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



The features available on this ribbon group are described below.



Select by Criteria

Click on the Select by Criteria button to open the [Select by Criteria Dialog](#). On oneline diagrams of even a modest size, this is an extremely useful feature and PowerWorld highly encourages you to use this feature.

Select Region

Use to select all objects in a particular region of the oneline. When the **Region Type** is **Rectangle** or **Ellipse**, then after clicking the button, click and hold the left mouse button on the oneline at the point where you would like to begin the selection. Then, drag the mouse to size the selection shape. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. When the Region Type is Polygon, then after clicking this button single click on your oneline diagram for each vertex of the polygon. Finally, finish the selection by double-clicking the mouse after which all objects inside your polygon will be selected.

Depending on the **Select Mode** below, every object located entirely **Inside** the region, or if you have chosen **Touching**, then every object inside or partially touched will be selected. Selected objects will have handles appear indicating that they have been selected.

Region Type

Specify a **Rectangle**, **Polygon**, or **Ellipse**. This controls the shape used to select objects on the oneline diagram when using the **Select Region** button above.

Select Mode

This combo box specifies how the **Select Region** button above works. Choose **Inside** in order to select only the objects completely inside the selection rectangle. Choose **Touching** in order to select every object either partially touched by or completely inside the selection rectangle.

There are also other ways to select multiple objects

- You can also select multiple objects manually by holding down the SHIFT key which clicking on objects in Edit Mode. This will also add to your selection.
- If you hold down the SHIFT and CTRL keys simultaneously and then click on your oneline diagram and select a region it has the same affect as click on the **Select Region** button and selecting a region on your oneline

Onelines Ribbon Tab

The Onelines ribbon tab generally consists of buttons and menus that provide access to modifying the appearance of a oneline diagram you have already created. For access to features regarding adding, moving, resizing, and formatting objects, see the Draw ribbon tab. The Oneline ribbon is shown below.



The Options ribbon consists of buttons and menus that provide access to customizing the software options. The ribbon is shown below.

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Active Ribbon Group

The Active ribbon group contains basic methods for customizing the appearance of basic analysis tools for customizing an existing oneline diagram. For more detailed help see the [Active](#) ribbon group topic

Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

General Options Ribbon Group

Dynamic Formatting Menu

This menu provides access to setting the [Dynamic Formatting Options](#) that apply to the active oneline, as well as formatting that applies to all onelines and Case Information Displays.



Custom Hint Values

Provides access to setting the [Custom Hint Values](#) that apply to all oneline objects..

Default Drawing Values

Provides access to the [Default Drawing Values](#).

Toggle Full Screen

Click this button to cause the presently active oneline to be toggled to full screen.

Keyboard Shortcuts

Click on this button to open the dialog for defining [Keyboard Shortcuts](#) which allow you to open a oneline or auxiliary file directly from a function key.

Views Ribbon Group

Bus View

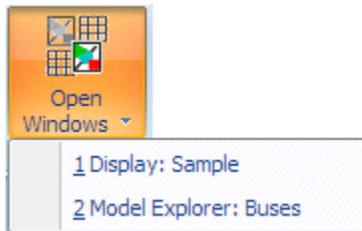
Opens the [Bus View Oneline](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Substation View

Opens the [Substation View Oneline](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that substation.

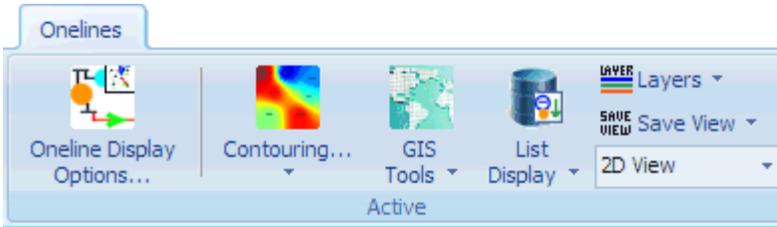
Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Oneline diagrams.



Active Ribbon Group

The Active ribbon group contains basic methods for customizing the appearance of basic analysis tools for customizing an existing oneline diagram.



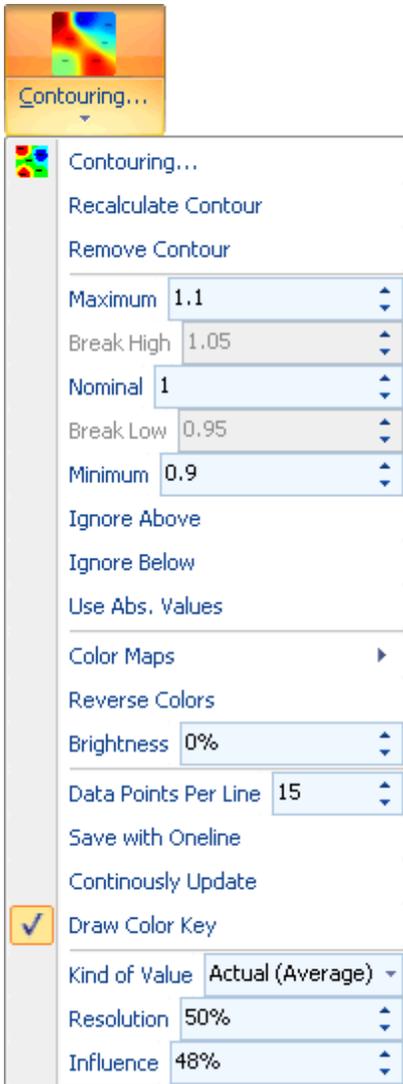
Online Display Options

Opens the [Online Display Options Dialog](#).

Contouring and Menu

Click the top part of the button to open the [Contouring Dialog](#).

Otherwise, click the drop-down to provide direct access to many of the Contouring Options.



GIS Tools Menu

Provides access to many Geographic Information System related tools that have been built into the software.



Insert GIS Data From Shapefile : Opens the [Shapefile Import dialog](#).

Export Oneline as Shapefile : Choose this to [Export the Oneline as a Shapefile](#).

Populate Lon,Lat with Display X,Y : Choose this to populate the Bus records latitude and longitude information based on the locations on the present oneline diagram. For more information see the [Populate help topic](#).

Closest Facilities to Point : Choose this to bring up the [Closest Facility to Point feature](#).

Great Circle Distance : Choose this to calculate the [Great Circle Distance between two points](#).

Insert Measure Line : Provides access to the [Insert Measure Line Feature](#).

Delete All Measure Lines : Provides access to the [Delete All Measure Lines feature](#).

List Displays Menu

Contains access to several ways of viewing the oneline objects inside of "[case information displays](#)". Of course instead of seeing case information, you instead see information about the oneline objects.



All Display Objects: Opens the [Display Explorer](#) providing access to all objects on the active oneline.

Unlinked Display Objects : Opens the [Unlinked Display Objects Dialog](#).

Only Selected Display Objects : Opens the [Display Explorer](#) providing access to objects on the active oneline, but defaults that explorer to only show the presently selected objects.

Browse Open Onelines: Brings up the [Browse Open Onelines dialog](#).

Display Objects Export Format Description : Opens a [Display Objects Export Format Description](#) dialog.

Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



Save View

Choose this option to open the [Save Views Dialog](#).

2D View / 3D View

Toggle between 2D and 3D to change the presently active online between the 2D and 3D modes.

Zoom Ribbon Group

The Zoom Ribbon Group is on both the [Onelines](#) ribbon tab and the [Draw](#) ribbon tab. To display large detailed power systems, Simulator's onelines possess zooming and panning capabilities. The Zoom Ribbon Group enables you to prescribe a zoom level either by directly specifying a zoom value or by selecting a rectangular region of the diagram on which to focus. There is also a button to open a dialog box from which you can select a bus on which to center the display. See [Zooming and Panning](#) for more information.

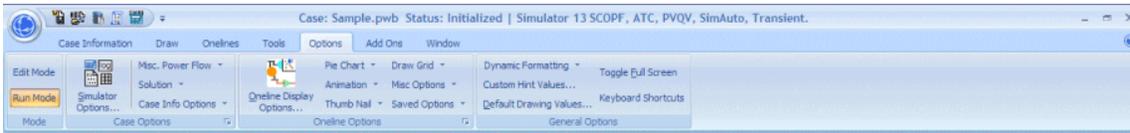


This toolbar contains the following several buttons for zooming the presently active oneline.

- **Zoom Area In /Out** : The top two buttons can be clicked to zoom or out on a particular area of the oneline. When you click these buttons, the cursor will change to a cross-hair waiting for you to choose the region to zoom into (or out of)
- **Zoom In / Out** : The middle two buttons can be clicked to change your mouse into a zooming tool. After choosing the button with a plus sign, each time you click on the oneline you will zoom in on that location. When choosing the button with a negative sign you will zoom out instead.
- **Present Zoom Level**: the number represents the present zoom level and can be manually changed to modify the zoom level.
- **Show Full** : Automatically zooms and pans the presently active oneline diagram so that all objects on the oneline diagram can be seen.
- **Find**: Provide access to the [Find Object on Online Dialog](#)

Options Ribbon Tab

The Options ribbon consists of buttons and menus that provide access to customizing the software options. All of the buttons on this ribbon are also available on one of the other ribbons, however this ribbon brings all the options in the software into one place. The ribbon is shown below.



The Options ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Case Options Ribbon Group

Simulator Options

Opens the [Simulator Options Dialog](#).

Misc. Power Flow Menu

Provides more direct access to some miscellaneous options that are also found on the Opens the [Simulator Options Dialog](#).

Solution Menu

Provides more direct access to some options related to the power flow solution algorithm. These options are all also available on the [Simulator Options Dialog](#).

Case Info Options Menu

Provides more direct access to some options related to the appearance of case information displays. These options are all also available on the [Case Information Display Options](#) on the [Simulator Options Dialog](#).

Online Options Ribbon Group

Online Display Options

Opens the [Online Display Options Dialog](#).

Pie Chart Menu

Provides more direct access to options related to Pie Charts. These options are all also available on the [Online Display Options Dialog](#) on the [Pie Charts/Gauges](#) page.

Animation Menu

Provides more direct access to options related to Animated Arrows. These options are all also available on the [Online Display Options Dialog](#) on the [Animated Flows](#) page.

Thumb Nail Menu

Provides more direct access to options related to Thumb Nail View. These options are all also available on the [Online Display Options Dialog](#) on the [ThumbNail View](#) page.

Draw Grid Menu

Provides more direct access to options related to the Draw Grid. These options are all also available on the [Online Display Options Dialog](#).

Misc. Options Menu

Provides more direct access to miscellaneous options related to oneline diagrams. These options are all also available on the [Online Display Options Dialog](#).

Saved Options Menu

This menu provides direct access to the for toggling between different sets of options. These different sets of options are defined on the [Online Display Options Dialog](#) by clicking on the **Save Options to Case** button on that dialog..

General Options Ribbon Group

Dynamic Formatting Menu

This menu provides access to setting the [Dynamic Formatting Options](#) that apply to the active oneline, as well as formatting that applies to all onelines and Case Information Displays..

Custom Hint Values

Provides access to setting the [Custom Hint Values](#) that apply to all oneline objects..

Default Drawing Values

Provides access to the [Default Drawing Values](#).

Toggle Full Screen

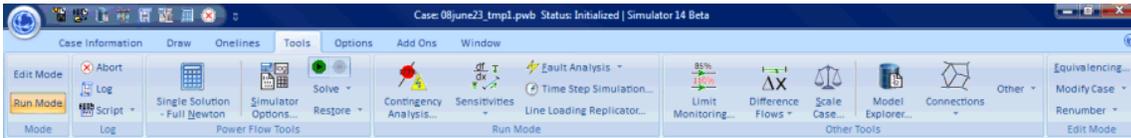
Click this button to cause the presently active oneline to be toggled to full screen.

Keyboard Shortcuts

Click on this button to open the dialog for defining [Keyboard Shortcuts](#) which allow you to open a oneline or auxiliary file directly from a function key.

Tools Ribbon Tab

The Tools ribbon consists of buttons and menus that provide access to the analysis tools that come with all versions of PowerWorld Simulator. Note: Features related to the various add-on tools including OPF, SCOPF, PVQV Curve, and ATC are found on the [Add Ons](#) ribbon tab. The tools ribbon is shown below



The Tools ribbon is broken into six ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Log Ribbon Group

Abort

Terminates the current Power Flow Solution. If the application is performing a timed simulation, pressing the abort button will pause the simulation. See [PowerWorld Simulation Control](#) for more details.

Log

Toggles the display of the [message log](#) window. The log window shows what is going on with the Power Flow Solution process and may prove useful when you are trying to track down a problem with a non-converging model.

Script

Opens the Script dialog, which can be used to call [script commands](#) or open [auxiliary files](#) containing script commands and data modifications. Note that the drop-down next to the Script button give convenient access to the [Quick Auxiliary Files](#).

Power Flow Tools Ribbon Group

This group represents the basic solution actions for solving the power flow.

For more detailed help see the [Power Flow Tools](#) ribbon group topic.

Run Mode Ribbon Group

The Run Mode Ribbon Group contains basic analysis tools that are only available in [Run Mode](#).

For more detailed help see the [Run Mode Ribbon Group](#) topic.

Other Tools Ribbon Group

The Other Tools Ribbon Group contains basic analysis tools that are normally available in both [Run Mode](#), and [Edit Mode](#).

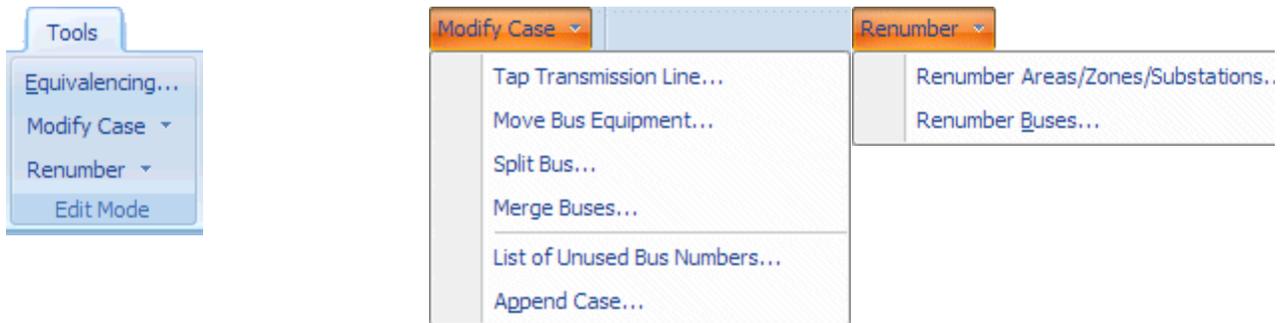
For more detailed help see the [Other Tools Ribbon Group](#) topic.

Edit Mode Ribbon Group

The Edit Mode ribbon group contains basic analysis tools that are only available in [Edit Mode](#). For more detailed help see the [Edit Mode Ribbon Group](#) topic.

Edit Mode Ribbon Group

The Edit Mode ribbon group contains basic analysis tools that are only available in [Edit Mode](#). The Group is shown below.



Equivalencing

Click this button to open the [Equivalencing Dialog](#).

Modify Case Menu

The Modify Case Menu contains actions related to modifying the network model.

Tap Transmission Line: Opens the [Tap Transmission Line Dialog](#).

Move Bus Equipment: Opens the [Move Bus Equipment Dialog](#).

Split Bus: Open the [Split Bus Dialog](#).

Merge Buses: Opens the [Merge Buses Dialog](#).

List of Unused Bus Numbers: Clicking this button to get a [list of unused bus numbers](#) in the model.

Append Case: Provides access to [opening a case while appending](#).

Renumber Menu

The Renumber Menu contains actions related to renumbering data in the model.

Renumber Areas/Zones/Substations: Opens the [Renumber Areas, Zones and Substations Dialog](#).

Renumber Buses: Opens the [Renumber Buses Dialog](#).

Other Tools Ribbon Group

The Other Tools ribbon group contains basic analysis tools that are normally available in both [Run Mode](#) and [Edit Mode](#). The group is shown below.



Limit Monitoring Settings

Opens the [Limit Monitoring Settings Dialog](#).

Difference Flows Menu



Difference Flows: Opens the [Difference Flows Dialog](#)

Present Topological Differences From Base Case: Opens the [Present Topological Differences From Base Case Dialog](#).

The remaining options in this menu all replicate the same behavior available on the [Difference Flows Dialog](#). For more details so that dialog.

Scale Case

Opens the [System Scaling Dialog](#).

Model Explorer

Opens the [Model Explorer](#) giving access to all the [case information displays](#) for the model.

Connections Menu

The Connection Menus contains actions related to analyzing the interconnectedness of the network model. These may be considered Graph Theory routines.



Determine Path Distances to Buses: Opens the [Determine Patch Distances from Buses or Group Dialog](#).

Determine Shortest Path Between: Opens the [Determine Shortest Path Between Dialog](#).

Facility Analysis: Open the [Facility Analysis Dialog](#).

Branches that Create Islands: Opens the [Branches that Create Islands Dialog](#).

Other Menu

The Other Menu contains other miscellaneous features.



Governor Power Flow: Opens the [Governor Power Flow Dialog](#).

Set Generator Part. Factors: Opens the [Set Generator Participation Factors Dialog](#)

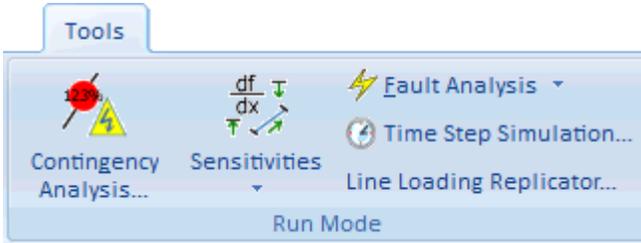
Set Selected Field for Network Cut: Opens the [Set Selected Field for Network Cut Dialog](#)

Set Selected From Selection: When a selection is chosen in [Edit Mode](#), click this to set the Selected? field to YES for that selection.

Charts Menu: The Charts menu is only available in [Run Mode](#). It provides access to various charts including the [Area ACE Chart](#), [Area Load and Generation Chart](#), [Area Losses Chart](#), [Area MW Transactions Chart](#), and [Area Average Cost Chart](#).

Run Mode Ribbon Group

The Run Mode ribbon group contains basic analysis tools that are only available in [Run Mode](#). The group is shown below.

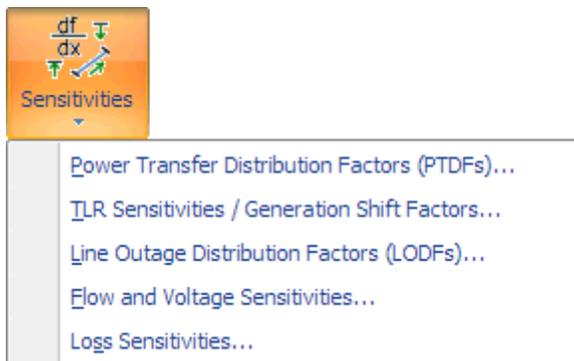


Contingency Analysis

Click this button open the [Contingency Analysis Dialog](#) providing access to the automated processing of a list of contingencies.

Sensitivities Menu

The Sensitivities Menu contains a list that provides access to all the sensitivity calculations available.



[Power Transfer Distribution Factors \(PTDFs\)](#) opens its associated dialog.

[TLR Sensitivities / Generation Shift Factors](#) opens its associated dialog.

[Flow and Voltage Sensitivities](#) opens its associated dialog.

[Loss Sensitivities](#) opens its associated dialog.

Fault Analysis

Click this button to open the [Fault Analysis Dialog](#).

Time Step Simulation

Click this button to open the [Time Step Simulation Dialog](#).

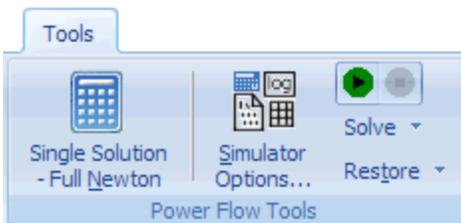
Line Loading Replicator

Click this button to open the [Line Loading Replicator Dialog](#).

Power Flow Tools Ribbon Group

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from either the [Quick Access Toolbar](#) or the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information.

The following tasks are also available from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab.



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Simulator Options

When choosing any of the solution methods, the options as specified in the [Simulator Options](#) will affect what calculation is performed. solves the power flow using either the Full Newton AC load flow or the DC Approximation load flow, as specified in the [Simulator Options](#).

Play Button

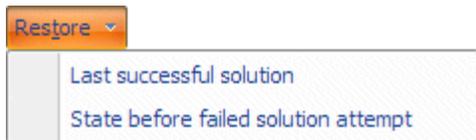
Click this button to cause repeated solutions of the system and to cause the animation of on-line diagrams

Pause/Stop Button

Click this button to stop the action of the Play button.

Restore Menu

The Restore menu offers option to either Restoring a Previous Solution or a Restore a Previous State



Sometimes a power flow attempt won't converge to a solution. When this occurs, the voltages and angles calculated by the solution engine will not satisfy the real and reactive power balance constraints at each bus. Then, the state currently stored in memory will not be an actual system operating point. It is often very difficult to coax the system to solve once it has failed to converge.

To help you recover from a solution attempt that has failed to converge (both timed simulations and single solutions), Simulator offers you two options.

After Simulator solves a system successfully, it will store the voltages and angles it found in memory. If the changes that you then make to the system result in a system that can't be solved, you can select **Restore > Last Successful Solution** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to reload the results of the last converged solution. After reloading this information, Simulator will re-solve the system and refresh all displays.

In addition to restoring the last converged solution, Simulator also gives you the ability to restore the state of the system as it was just prior to the unsuccessful solution attempt. This can be thought of as "un-doing" the effect of the solution attempt. Before attempting a solution, Simulator stores the state of the system in memory. If it solves the power flow successful, Simulator will discard this pre-solution state. However, if the power flow fails to converge, Simulator will keep the state in memory. To recover it, select **Restore > State Before Solution Attempt** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Simulator will replace the non-converged post-solution state with the pre-solution state and refresh the displays. You can then play with the system to try to make it easier to solve.

If you are working with large systems, you should be aware that saving these system states can consume a lot of memory. Therefore, Simulator offers you the option to disable one of both of these features. To do this, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab to open the [Simulator Options Dialog](#), and then go to the Storage sub-tab of the Solution tab. You will see two checkboxes there that can be modified to control whether or not these extra system states are saved.

Solve Menu



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab, or from the [Quick Access Toolbar](#). This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Polar NR Power Flow

This performs a Newton Raphson power flow solution using polar coordinates instead of rectangular coordinates as is normally done in Simulator.

DC Power Flow

This automatically changes the Simulator Options to use the [DC Approximation](#) and then immediately performs the DC power flow solution.

Reset to Flat Start

Select **Solve > Reset to Flat Start** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to initialize the Power Flow Solution to a "flat start." A flat start sets all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero.

Usually, a flat start should be used only if the power flow is having problems converging. You can also use the flat start option on the [Simulator Options Dialog](#) to initialize every solution from a flat start.

Robust Solution Process

The Robust Solution Process provides a method to attempt to reach a solution when the standard load flow (Newton-Raphson) solution fails. The robust process performs a solution in a series of steps.

First, the robust solution will turn off all controls in the case. Then the load flow will be solved using a fast decoupled power flow. If the fast decoupled power flow reaches a solution, Simulator then immediately solves the load flow using the Newton-Raphson load flow, still keeping the controls turned off. If the Newton-Raphson solution is also successful, Simulator will begin adding controls back into the solution process, one type of control at a time. Thus the generator MVAR controls are added back in, and the load flow is resolved. Then the switched shunt controls are restored, and the load flow is again resolved. Simulator will continue in this manner by reintroducing next the LTC control, followed by the area interchange control, and lastly the phase shifter control. Furthermore, when reintroducing the phase shifter control, the controls are added one at a time for each phase shifter, with a load flow solution occurring after each.

Window Ribbon Tab

The Window ribbon consists of buttons and menus that provide access to customizing the windows in the user interface. It also provides access to help topics and topics regarding the Auxiliary File Format. The ribbon is shown below.



The Case Information ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

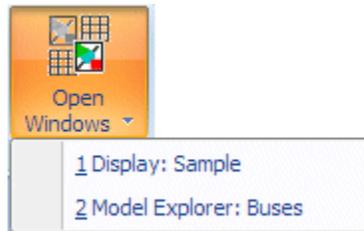
Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Window Ribbon Group

Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



Switch to Free-Floating Windows (Switch to Standard Container Window)

Choosing this option toggles the program from between two different window environment

1. All diagrams and dialogs are contained within the Simulator program shell
2. All diagrams and dialogs are free-floating on your monitor.

For more information see the [Window Basics](#).

Reset to Defaults

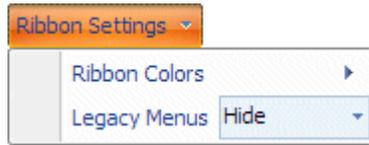
The program keeps track of dialog Free-Floating/Contained Style, Size, and Position. Clicking this button will reset all these to the factory default settings. Nearly all dialogs in will then be reset back to Contained with a default size and position as well. For more information on Style, Size and Position, see [Window Basics](#).

Refresh Displays

Redraws (refreshes) each of the open windows. Simulator usually automatically refreshes the open windows as necessary. However, this action allows you to trigger the refresh when you want it.

Ribbon Settings Menu

This menu contains options for customizing your ribbon.



Ribbon Colors : You can change the color scheme to either **Blue**, **Silver** or **Black**

Legacy Menus : Choose to Hide or Show a special Menus ribbon tab. The Menus ribbon tab mimics the menu and toolbar structure that existed in older versions of Simulator.

Arrange Windows

The options under Arrange Windows are Tile Horizontally, Tile Vertically, and Cascade. Tile rearranges the open oneline diagrams such that the total window area is divided equally among all of them and each is completely visible. Cascade Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

Toggle Full Screen

Toggle full screen makes the currently selected diagram window switch to full screen mode. Full screen mode will dedicate the screen to the window, with all other windows, including the Simulator menus and toolbars, to be hidden. To get back to normal mode, right-click on the diagram in full screen mode and select Toggle Full Screen from the popup menu.

Help Ribbon Group

Contents

Opens the [Simulator Options Dialog](#).

Set Help File

Opens the [Case Description Dialog](#).

About

Click this to open the About dialog. The About dialog contains information about the software as well as showing you the Build Date of your version of Simulator. PowerWorld release patch version of Simulator on our website. These patch versions are named by date and is shown on the About dialog.

There is also an option on the About dialog to Change License Key. This allows selection of a new license key file. Add ons are controlled through the license key. If you are using different versions of Simulator with different add ons, this option provides the mechanism for switching between the different versions of Simulator. You will need a different license key file if you are using different licenses for Simulator that have different add ons.

PowerWorld Website

Click this to open your default web browser to the PowerWorld Corporation Website.

Check for Updates

This button will open the PowerWorld website to the patch download page. You will be required to enter your Account Username and Product ID (password) to check for and download patches for your version of Simulator.

Auxiliary File Ribbon Group

Load Auxiliary

Choose this option to load an [Auxiliary File](#).

Load Display Auxiliary File

Choose this option to load a [Display Auxiliary File](#) to append to the active oneline.

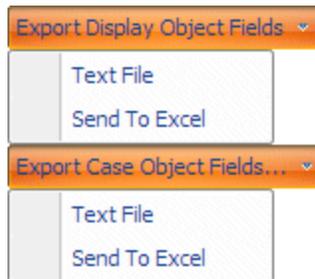
Auxiliary File Format

Opens the [Bus View Oneline](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Export Display Object Fields Menu

Export Case Object Fields Menu

Choose these options to export a list of fields for each type of object in a case (or a Oneline Display). The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.



Chapter 2: Using Oneline Diagrams

Simulator has been designed to be both highly graphical and interactive. The key to making effective use of Simulator lies in understanding the oneline diagrams. This chapter provides essential information on how to use and customize onelines.

The following material is included:

- Oneline Diagram Overview
- Oneline Tools and Options
- Printing Oneline Diagrams
- Relationship Between Display Objects and the Power System Model
- GIS Tools

Oneline Diagram Overview

The purpose of the oneline diagram is to show information about the power system graphically. Such displays are called oneline diagrams (onelines) because the actual three-phase power system components are represented using a single line. Simulator onelines "come alive" via:

- [Animation](#)
- [Contouring](#)
- [Zooming and panning](#) capability and
- [Conditional display of objects](#)

Additionally, a key aspect of Simulator is the ease with which it allows the user to examine and modify many of the shown on the oneline diagram.

The user may open any number of oneline diagrams, including multiple copies of the same oneline.

Relationship Between Display Objects and the Power System Model

Display Object:

An item shown on a oneline diagram. Display objects typically have an associated model object. Examples include buses, transmission lines, transformers, generators and loads. Display objects not associated with a model object are called unlinked objects.

Model Object:

A power system element contained in a case.

A key strength of the Simulator is its ability to allow users to manipulate a power system model graphically. This capability greatly simplifies the work involved in developing or maintaining a power system case for both novice and advanced users. However, it is important to keep in mind the distinction between the shown on the onelines and the actual power system model, consisting of model objects. A key concept is that any number of display objects, including none at all, can be associated with a single model element.

Simulator uses a bus-oriented model. In other words, the model objects are either the buses themselves, objects that are radially attached to a bus (i.e., loads, generators and switched shunts), or objects that join two buses (i.e., transmission lines, transformers or dc lines). As long as there is a one-to-one mapping between display objects and model objects, the distinction between the two could be made entirely transparent to the user.

It is reasonable, and often quite useful, to use more than one display object to represent a single model object. For example, by using the [Conditional display of objects](#) feature, two bus display objects could be used on a single oneline to represent the same bus. One bus might be visible over a particular zoom range, while another, with perhaps a different size/thickness, is visible over another range. Alternatively, the same bus could be represented using display objects drawn on separate onelines.

An ambiguity arises when the user uses the Cut command to delete an object. Is he or she deleting just the Display Object or both the Display Object and the Model Object? To alleviate the problem, Simulator prompts you when you are deleting a display object with an associated model object to delete both the Display Object and its associated model object record, delete just the display object, or cancel the delete.

In addition, there is no requirement that model objects have a corresponding display object. Thus, you could use the oneline diagram to show just a fraction of the total system buses and other devices. You can use the [Case Information](#) menu to view the model objects directly regardless of whether or not they are shown on a oneline.

Oneline Local Menu

The local menu provides access to a number of options and tools directly from the oneline. To display the local menu, position the cursor on an empty portion of the oneline then click the right mouse button. While most options are always available on the popup menu, a few options are only available on the popup menu in run mode.

Find Object on Oneline (Edit and Run Modes)

Displays the Find Object on Oneline tab of the [Zoom, Pan, and Find Objects](#) dialog.

Oneline Display Options (Edit and Run Modes)

Displays the [Oneline Display Options](#) dialog. This dialog allows you to customize the appearance of the oneline.

Pan/Zoom Control (Edit and Run Modes)

Displays the Zoom/Pan tab of the [Zoom, Pan, and Find Objects](#) dialog.

Edit Screen Layers (Edit and Run Modes)

Opens the [Screen Layers case information display](#).

Show Layer (Edit and Run Modes)

Allows activation of any saved screen layer.

Save/Edit/Delete View (Edit and Run Modes)

Displays the [Save View Level Dialog](#). This dialog is used to set defined locations on the oneline for recalling specific views from a list of saved views.

Go To View (Edit and Run Modes)

Allows the user to go to a specific location and zoom level on the oneline by choosing from a list of saved views. This option does nothing if no views are saved.

Contouring (Run Mode Only)

Displays the [Contour Options Dialog](#). This dialog allows you to contour the system voltage magnitudes or angles.

Difference Flows

Displays the [Difference Flows Dialog](#). This dialog is used to compare two power system operating points.

Dynamic Formatting (Active Oneline) (Edit and Run Modes in Oneline Diagrams)

Displays the [Dynamic Formatting Dialog](#). This dialog is used to change the rendering of objects in oneline diagrams according to the state of the represented object in the power system.

Dynamic Formatting (All Views) (Edit and Run Modes in Bus and Substation Views)

Displays the [Dynamic Formatting Dialog](#). This dialog is used to change the rendering of objects in bus and substation views according to the state of the represented object in the power system.

All Display Objects (Edit and Run Modes)

Opens the [Display Object case information display](#), showing all the objects contained in the oneline diagram.

Only Selected Display Objects (Edit Mode Only)

Opens the [Display Object case information display](#), showing only the currently selected objects in the oneline diagram, by automatically checking the **Show Only Objects Selected on Display** box.

Load Display Auxiliary File (Edit and Run Modes)

Loads a [display auxiliary file](#).

Print Oonline (Edit and Run Modes)

Sends a copy of the oneline diagram to the printer. Selecting this option has the same affect as selecting **File, Print Oonline** from the main menu. See [Printing Oonline Diagrams](#) for more information.

Copy Image to Clipboard (Edit and Run Modes)

Copies the oneline file into the Windows clipboard using the Windows Metafile format (*.wmf). You can then paste the oneline into another program, such as a word processor. See [Copying Onelines to Other Programs](#) for details.

Export Image to File (Edit and Run Modes)

Saves a copy of the entire oneline diagram image to a file. Oonline image files can be saved in bitmap (*.BMP), Windows Metafile (*.WMF, .EMF), or JPEG format (.jpg).

Toggle Full Screen (Run Mode Only)

Sets the oneline diagram display window to full screen size. This option hides all toolbars and status bars. To return the window to normal, right-click on the oneline diagram and un-check the Full Screen option.

Embed a Display (Run Mode Only)

Clicking on this option allows you to open another oneline diagram (or the same one) and embed the new display inside the existing display. You can choose what percentage size the embedded display should be, and where within the existing window the embedded window should be placed.

Form Control > (Edit and Run Modes)

Options on this submenu allow you to resize the window, shift the window, or close the window.

Oneline Display Options Dialog

The Oneline Display Options dialog allows you to customize the display of the oneline diagram. You can access this display by right-clicking anywhere on an empty portion of a oneline and selecting **Oneline Display Options** from the resulting [oneline local menu](#).

This dialog can also be display by going to the [Options](#) ribbon tab and choosing **Oneline Display Options** from the **Oneline Options** ribbon group.

This dialog houses several tabbed pages that govern various aspects of the oneline display. See [Animated Flows](#), [Display Object Options](#), [Display Options](#), [Grid/Highlight Unlinked Objects](#), [Geography/Coordinates](#), [Memo](#), [Movie](#), [Pie Charts / Gauges](#), [Thumbnail View](#), or [Substations](#) for information on the respective tabs.

Save Options to Case

Of important note is the general option at the bottom of this dialog labeled **Save Options to Case**. What this allows you to do is define a set of oneline options in the dialog, and then save that definition of options *in the case* by giving the definition a name. Once you have named that set of options, you can recall it later by coming back to this dialog and choosing the name from the list of option set names that appears near the bottom of the dialog. You can define as many different sets of custom options as you wish. You can also use an option set saved with the case when you create a oneline view using the [Save Views](#) dialog.

Also note that once you have defined these sets of Saved Oneline Options, you can then quickly apply them to the active oneline diagram by using the **Saved Options Menu** on the [Options](#) ribbon tab.

Display Options

Display Detail

The Display Detail box allows you to control how much information is shown on the oneline display. This option is available only in run mode. All elements on a oneline diagram are automatically visible in edit mode. There are four choices for run mode detail filtering:

Minimal Show the oneline background, branch circuit breakers, generator MW output, and load MW/MVR.

Moderate Show all Minimal information, along with bus voltages and all line flow pie charts.

Complete Show all information.

Custom Selecting this option and then clicking the **Set...** button opens the Custom Detail Dialog, which allows you to customize the oneline diagram to hide objects that do not meet your desired specifications. This dialog looks and works very similar to the [Select By Criteria](#) tool, allowing you to choose objects filtered by area, zone, voltage, and screen layer.

Of course, in order for a certain display object to appear, it must have been placed there by the person who designed the oneline.

Emphasize Specific Objects

This option allows you to choose to "emphasize" desired elements on a oneline diagram. The emphasis is in the form of the emphasized elements being in full color, with de-emphasized elements being muted colors.

To choose emphasis of certain elements, first check the box labeled **Do Emphasis**. When you first check this box, it will open the Emphasis Filter dialog, which looks and works very similar to the [Select By Criteria](#) tool. You can choose the oneline elements you wish to emphasize, and include filtering by area, zone, voltage, and screen layer.

The degree of emphasis (muting of de-emphasized elements) can be controlled by the **Emphasis Amount** slide bar.

Background Color

Select **Change Background Color** to select a different color for the oneline diagram background color. Select the **Set as Default Background Color** option to set the background color as the default for all oneline diagrams.

Use Absolute Values for MW Line Flows

If checked, this option will cause all MW flow text fields for lines to be displayed as the absolute value of the flow. Otherwise, the MW flow text fields will be positive near the source end of the line and negative near the sink end of the line.

Use Absolute Values for Mvar Line Flows

If checked, this option will cause all Mvar flow text fields for lines to be displayed as the absolute value of the flow. Otherwise, the Mvar flow text fields will be positive near the source end of the line and negative near the sink end of the line.

Use Absolute Values for MW Interface Flows

If checked, this option will cause all MW flow text fields for interfaces to be displayed as the absolute value of the flow. Otherwise, the MW flow text fields will be positive near the source end of the interface and negative near the sink end of the interface.

Enable Mouse Wheel Zooming

When this box is checked, zooming can be done with a mouse wheel. The default is off.

Show Oneline Hints

When this box is checked, hovering the cursor over an object will briefly display information about the object in a popup hint.

Visualizing Out-of-Service Elements

Out-of-service elements can have optional visualization settings that makes them "stand out" in relation to in-service devices on the oneline diagram. The options for out-of-service devices includes **Use Dashed Lines**, **Draw and X Through Off-Line Generators**, and **Blink**. Additional controls for the Blink option are available, for setting the blink interval and color. Any combination of these options can be used for out-of-service elements, although the **X** option only applies to out-of-service generators.

Browsing Path for Oneline Diagrams

This option applies when you have [Oneline Links](#) included on a oneline diagram. Rather than specify the full path and name of a oneline diagram as a oneline link, you can specify the file name only. When the link is clicked in Run Mode, Simulator will check all directories listed here, in order, to try and find the oneline file name stored with the link.

Pie Chart/Gauge Options

Show Pie Charts/Gauges in Run Mode

When this option is checked, the pie charts and line gauges will be visible during Run Mode. Otherwise, they will be visible only during Edit Mode.

Only Show Pie Charts/Gauges if relevant data exists

The relevancy of pie charts and gauges depends on the style of the pie chart or gauge. For MVA, MW, Mvar and Amp styles, the pie charts are considered irrelevant if the limits on the line or interface are 0. For max percent loading under contingency, the pie chart is irrelevant if no violation(s) under contingency occurred on the element. For the PTDF style, the pie chart is invalid if the PTDF value has not been calculated for the line or interface.

The options for pie charts and gauges are split into four sections, discussed in the additional topics:

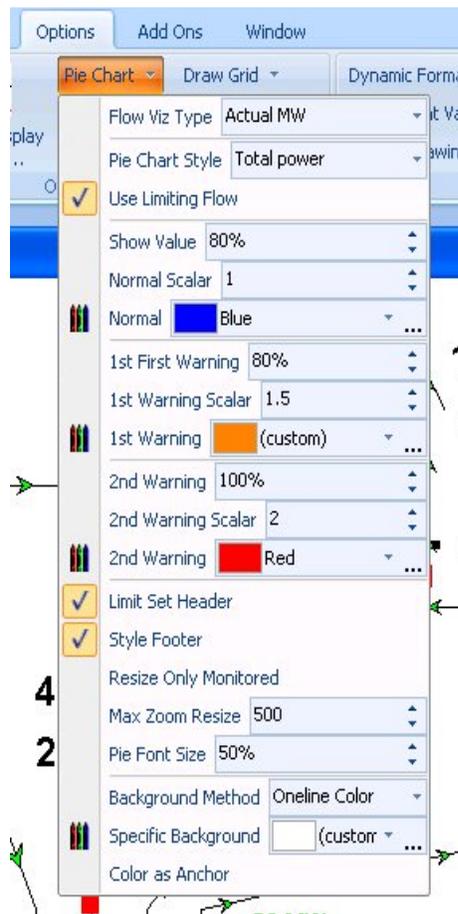
[Pie Charts/Gauges: Lines](#)

[Pie Charts/Gauges: Interfaces](#)

[Pie Charts/Gauges: Pie Chart/Gauge Styles](#)

[Pie Charts/Gauges: General Options](#)

Many of these same options are available on the Pie Chart menu found on the [Options ribbon tab](#).



Animated Flows Options

The fields on the Animated Flows page of the [Online Display Options](#) dialog are used to customize the appearance of the animated flows on the oneline diagram.

Show Flow Symbols

Determines whether power flows are animated on the onelines. If this option is not checked, then no flow symbols appear on the oneline.

Show Flows On

Allows you to choose whether or not to display animated flows individually on Branch, Load, Generator and Shunt objects. Uncheck the box(es) for the objects that you do not wish to display animated flows.

Base Flow Scaling on

Animated flows on onelines may depict either actual power flows or [power transfer distribution factors](#), depending on what you choose here. You can also choose to animate both the MW and MVAR flows or the MW and PTDF flows simultaneously. The Custom Float 1 option allows user specification of a value to depict in the flow animation. To use this option, values should be entered in the Custom Floating Point 1 field for the appropriate devices.

Animate

This option determines whether or not the flow arrows are mobile or stationary on the oneline diagram. Unchecking this box will cause the flow arrows to remain stationary on the diagram.

Scale Speed of Flow

Checking this box will cause the arrows to flow at a speed proportional to the represented quantity. When it is not checked, all of the flows will be at the same speed.

Scale Size of Flow

If checked, the size of the animation symbols will vary to represent the magnitude of flow on the element (respective of the **Animation Parameter**.) Otherwise, the symbol size will be the same on all devices regardless of the magnitude represented.

Reset Animated Flow Offsets

This button allows the user to reset the animated flows to start at a specified offset position. Mostly this would be used to reset the offset to 0, which would cause the animated flows to start at the beginning of the line or element. Since the animation moves the flow arrows every time the load flow is resolved, resetting flows to a specific offset could be useful for comparing different load flow solutions by looking at differences in the animated flow objects.

Set Size, Density, and Reference Values for this oneline

Simulator detects the current zoom level and object parameters of the selected oneline diagram, and automatically adjusts the animation settings in an attempt to optimize the animation quality.

Size

Determines the relative size of the animated flows on the devices. Increasing this number increases the size of the flow symbols. This field may range from 1 to 999.

Density

Determines the relative density of the animated flows on the devices. Increasing this value causes Simulator to display a greater number of flow symbols per unit distance on the oneline. This value may range from 1 to 999.

Scaling Based on

Determines whether the size and speed of animated flows represent actual flow or percentage loading. This option applies only to transmission lines and transformers as flows on other devices, such as loads and generators, always represent actual flow.

Max. Zoom Level to Scale Size

As a oneline is zoomed, the animated flow symbols increase in size. The value of Max. Zoom Level to Scale Size caps the size of the animated flows so that zooming beyond this level results in no further increase to the size of the flow symbols.

Maintain Density above Maximum Zoom Level

As a oneline is zoomed, the animated flow symbols decrease in density. Checking this box maintains the density of the animated flows after the maximum zoom level has been reached.

Minimum Size in Pixels for In-service Elements

Allows the specification of a minimum size for the animated flow objects for power systems elements that are in-service. The flow objects are sized based on the flow through the element. For elements with very small flows, it is often difficult to see the flow objects, and the minimum size can be set to make these flow objects more visible. The larger the minimum size value becomes, the more uniform the flow objects will become as elements are no longer sized based on their actual flow but rather the minimum size.

Reference Values for Scaling

Maximum flow reference for sizing the animated flows. The lower the MVA reference, the larger the animated flows appear on the oneline diagram as the actual flow value is compared to the reference value for scaling.

Symbol Shape

Simulator can display animated flows using directional arrows, circles, squares, or triangles.

Animation Rate

If the oneline animation is too fast or too slow, the animation rate can be adjusted by moving the slide bar.

Symbol Fill Color

Shows the fill color used for the animated flows if **Use Fill Color** is checked. Double-click on these fields to change the colors of the six different types of animated flows.

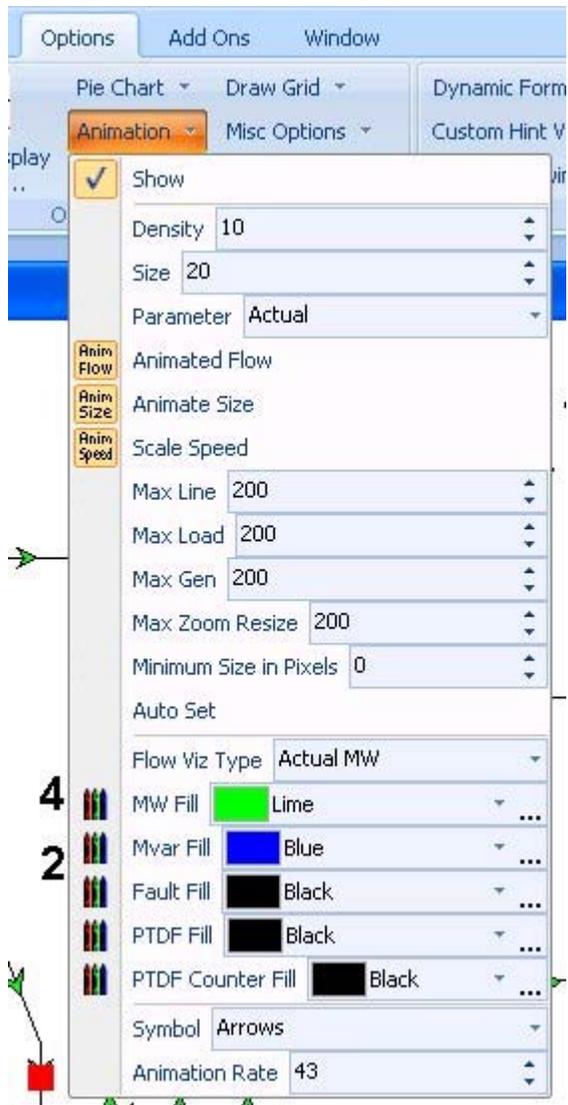
Use Fill Color

Checked if animated flow symbols should be filled using the **Fill Color**.

Show PTDF Counter Flow

Check this option if you wish for PTDF values that are in the opposite direction of the actual flow to be displayed using a different symbol color when visualizing PTDFs.

Many of these same options are available on the Animation Menu found on the [Options ribbon tab](#).



Thumbnail View

The thumbnail view allows the user to see an overview of the oneline diagram in a smaller window in a specified location on the oneline diagram. The thumbnail view is useful when the user has zoomed in to a specific area of the oneline diagram, but still wants to see what part of the system they are observing on a larger scale as they pan around the diagram. The other application is to observe a more detailed part of the system in the thumbnail view as the user moves the cursor over the oneline diagram. The thumbnail view is not visible by default, but can be set up and displayed from the [Oneline Display Options](#) dialog. The options for the thumbnail view are:

Show ThumbNail View

If checked, the thumbnail view will be visible in the specified corner of the oneline diagram.

Size of ThumbNail View

The size of the thumbnail view as a percentage of the size of the oneline diagram.

Zoom level of Thumbnail View

The amount to multiply the oneline diagram zoom level for display in the thumbnail view. The higher the multiplier, the more of the diagram you will see in the thumbnail view. You can instead indicate that the thumbnail view should display the diagram at a specified zoom level rather than a dynamically scaled zoom level based on the current diagram's zoom level.

Location

Choose the location on the oneline diagram in which the thumbnail view is to appear.

Border Width in Pixels

The pixel thickness of the border around the thumbnail view.

Background Color

If **Use One-line Background Color** is checked, the thumbnail background color will be the same as the oneline diagram. If it is not checked, then a different thumbnail background color can be selected by clicking on the colored box next to the **Use Custom Background Color** label.

Thumbnail view follows mouse cursor

This setting is very useful when is combined with the Zoom Level. Using a low multiplier and checking the Thumbnail view follows mouse cursor box will show a more detailed part of the system over which the cursor is being moved.

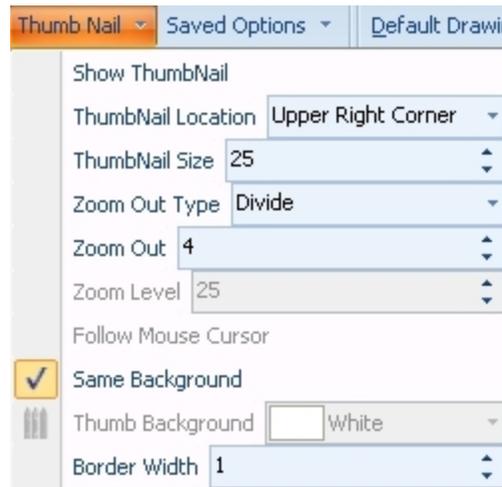
Percent Transparent

This setting allows you to make the thumbnail window completely opaque, completely transparent, or some measure in between. The more transparent the thumbnail window, the more detail of the underlying oneline diagram is visible through the thumbnail window.

Many of these same options are available on the [Thumb Nail](#) options menu found on the [Options](#) ribbon tab.

Thumb Nail Menu

The Thumbnail View Options quick menu provides access to most of the options contained in the [Thumbnail View Options](#) dialog in an individual manner through menu commands. Select the **Thumb Nail** option from the **Online Options** ribbon group on the [Options](#) ribbon tab to access the menu.



This quick menu contains the following default buttons, in order from top to bottom:

- Show thumbnail on the screen
- Thumbnail location on the screen
- Size of Thumbnail View, as a percentage of the window size
- Zoom out type (divide or zoom level)
- Zoom out factor, defining the ratio of the diagram seen in the thumbnail relative to what is visible in the diagram itself.
- Enable/Disable Thumbnail View to follow mouse cursor
- Enable/Disable Using the Same Background as the main window
- Background Color, if not the same as the main window
- Line width of the border around the thumbnail window

Display Object Options

General Options (All) Tab

Percent of Injection Group/Owner object height used by name

This option allows the user to specify the height of the name with respect to the rectangle of the injection group graphical objects.

Display Circuit Breakers in Generators, Loads and Switched Shunts

Uncheck this box to hide circuit breakers from appearing in generators, loads, and switched shunts. If this option is checked, the user can use in **Display breakers on...** to select individually what elements can display circuit breakers.

Display Rotors in Generators

Uncheck this box to hide the rotors of the generators.

Change in Gen Rotor Angle per Refresh (degrees)

This value specifies the change in the rotor angle every time the animated flows refresh. This means that the greater the angle value, the faster the rotor will seem to rotate when the oneline is animated. Entering a negative number will cause the rotor to spin counter-clockwise.

Show Field Inside Generator

Fields can be shown inside generator objects. Select a field from the drop-down box or use the Find button to search through the list of fields available for a generator. Set the field to *No Field* to not display any field inside generator objects. Set the **Digits** and **Decimals** for floating point fields. Adjust the **Relative Font Size** to change the size of the font relative to the generator object size. To ensure that the field fits entirely inside each generator object, check the option to **Dynamically decrease the font size so that the field fits inside the generator**.

Multi-Section Line Display Objects

Intermediate Bus Rotation Angle

Normally, the intermediate buses are perpendicular to the line. The value entered here can rotate the intermediate bus with respect to the original position. Therefore, a value of 0 degrees, will keep the intermediate buses perpendicular to the line, while a value of 90 degrees will set the intermediate buses aligned to the line.

Intermediate Bus Relative Size

This option is actually a multiplier for the size of the intermediate buses specified for each multi-section display object.

Bus Field Voltage Option

This option allows the user to specify in which units the bus fields with *Bus Voltage* as type of field will be displayed. This setting is only valid when the type of field selected is *Bus Voltage* in the [Bus Field Information](#) dialog. Choosing a voltage field in the Select Field will return the voltage value in the specific units of the field.

Circuit Breakers Tab

Circuit Breaker Display Objects > Normally Closed

This section is used to specify the colors for all the circuit breakers normally closed in the oneline diagram. The colors that can be modified are the border and fill colors for presently closed and open circuit breakers. In order to change a color, click on the color box or on the **Change** button. Check **Use Fill** to use the specified fill color.

Circuit Breaker Display Objects > Normally Open

This section is used to specify the colors for all the circuit breakers normally open in the oneline diagram. The colors that can be modified are the border and fill colors for presently closed and open circuit breakers. In order to change a color, click on the color box or on the **Change** button. Check **Use Fill** to use the specified fill color.

Shape

Specify the shape of circuit breakers on transmission lines and transformers. The breakers shown on loads, generators, and switched shunts will always be the default rectangle shape.

Substation Display Options

The **Substations** tab of the [Oneline Display Options](#) dialog provides parameters for customizing the appearance of [Substation](#) display objects. This tab is only available if the case contains at least one substation.

The upper right-hand corner of the Substation page displays a template of the substation object's appearance in terms of locations or "zones" where information can be displayed on the object. The rest of the page can be used to describe the appearance of the object and the information that appears in the substation objects.

Upper % of Height

This percentage indicates how much of the object is populated by the "upper" zone of the object. This includes the *Upper Field*, *Upper Left (UL)* and *Upper Right (UR)*. As you decrease this percentage, the upper zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the upper zone.

Identifier % of Height

This is the percentage of the height of the substation object that is occupied by the identifier or "middle" zone. This is the substation identifier section of the object.

Lower % of Height

This is the percentage of the substation object that is occupied by the "lower" zone of the object. This includes the *Lower Field*, *Lower Left (LL)* and *Lower Right (LR)*. Unlike the previous two percentages, this percentage cannot be directly modified. Instead, it is automatically determined based on the settings of the *Upper* and *Identifier* percentages.

Left % of Width

This percentage indicates how much of the object is populated by the "left" zone of the object. This includes the *Upper Left (UL)* and *Lower Left (LL)*. As you decrease this percentage, the left zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the left zone.

Identifier % of Width

This is the percentage of the width of the substation object that is occupied by the identifier or "middle" zone. This is the substation identifier section of the object.

Right % of Width

This percentage indicates how much of the object is populated by the "right" zone of the object. This includes the *Upper Right (UR)* and *Lower Right (LR)*. As you decrease this percentage, the right zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the right zone.

Buffer Percent

The buffer width for the height settings provide a buffer zone of the percentage width specified horizontally between the Identifier section and the upper / lower field sections. Similarly, the buffer width for the left and right width fields indicates a percentage buffer vertically between the upper / lower field zones and the four corner zones.

Substation Identifier

Choose how the identifier will be displayed for the substation. Choose to display by name, by number, or by combinations of name and number. The identifier will be displayed in the *Identifier* or "middle" zone of the substation object.

Shape

Select the shape used for substations that are not overriding the [Substation Layout Settings](#).

What should be done when identifier text does not fit inside the width

As the name of this option suggest, you can choose how the text in the identifier section of the object should be modified if resizing of the substation object causes the text to be too large for the modified size.

Extra Substation Fields

These four fields, two of which are represented as the *Upper Field* and *Lower Field* on the template, can be customized to display any substation field of your choosing. Use the drop-down arrow of the appropriate box to choose a field, or use the Find button to pull up a list of fields to choose from. Once the field has been chosen, set the digits and number of decimal places for the field.

Upper-Left Symbol (UL)

Choose what symbol to display in the upper-left zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Upper-Left location of the object to indicate such a presence. By default, the Upper-Left zone will display the Generator symbol.

Upper-Right Symbol (UR)

Choose what symbol to display in the upper-right zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Upper-Right location of the object to indicate such a presence. By default, the Upper-Right zone will display the Load symbol.

Lower Left Symbol (LL)

Choose what symbol to display in the lower-left zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Lower-Left location of the object to indicate such a presence. By default, the Lower-Left zone will display the number of buses in the substation.

Lower-Right Symbol (LR)

Choose what symbol to display in the lower-right zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Lower-Right location of the object to indicate such a presence. By default, the Lower-Right zone will display the Switched Shunt symbol.

Oneline Animation

An important feature of PowerWorld Simulator is its support of animated onelines. The use of efficient display algorithms allow animation rates that are typically greater than several times per second, even on large cases and on onelines with a significant number of objects. The extensive use of animation makes the display "come alive" so that system conditions can be ascertained more easily.

In Simulator, animation is started from Run Mode by selecting the **Play** button in the **Power Flow Tools** ribbon group on the **Tools** ribbon tab. In Viewer, animation is started automatically when you load a case.

The animation can be controlled and customized from the **Animated Flows Tab** of the **Oneline Display Options** dialog. To access this dialog, select **Oneline Display Options** from the **Oneline Options** ribbon group on the **Options** ribbon tab, or right-click on an empty area of the oneline diagram and select **Oneline Display Options** from the resulting **local menu**.

Copying Onelines to Other Programs

The onelines can be easily copied to other programs. This allows you to add PowerWorld onelines to your word processor documents or slide presentations. The simplest way to copy a oneline diagram to another program is to use the Windows Clipboard. This is accomplished as follows:

- In Simulator or Viewer, zoom and/or pan the display to the portion of the oneline diagram you would like to copy.
- Right-click on an empty portion of the oneline to display the [local menu](#).
- Select **Copy Image to Clipboard** menu item. This places a copy of the oneline into the Window's clipboard.
- In the other program, use **Paste** or **Paste Special** to copy the contents of the clipboard into that program. The oneline is pasted into the program using the Metafile format.

In addition, oneline diagrams can also be [saved as image files](#) by right-clicking on an empty portion of the diagram and choosing **Export Image to File**.

Display Explorer

The Display Objects case information display presents data describing each graphical object in the oneline diagram. This display functions in the exact same manner as the [Model Explorer](#) and is thus called the **Display Explorer** instead. The Display Explorer contains [Case Information Displays](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its records as well as view the information dialog of its associated power system object if it has any. You can also sort the object records by clicking on the heading of the field by which you want to sort. Additionally, when Simulator is in [Edit Mode](#), the toolbar **Records** menu allows you to remove existing objects from the oneline diagram.

To show the Display Objects case information display, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group select **List Display > All Display Objects**.

As mentioned above, the Display Explorer functions identically to the [Model Explorer](#). The only differences in functionality are due to additional options found on the bottom left of the dialog. This offers the following additional features.

Show Only Objects Selected

By checking this box, only the selected display objects currently selected in the oneline diagram will be shown.

How to List Grouped Objects

This specifies how to list grouped objects if there are any in the oneline diagram. The choices are to list individually the components of the group, to list only the groups, or to list both the individual elements and the groups. If there are not grouped objects in the oneline diagram, this control is disabled.

Save Complete Display to AXD

By clicking this button, all the objects that are shown in the case information display will be saved to a [display auxiliary file](#).

By default, the display object records case information display contains the following fields:

Type

The type of display object.

X/Longitude Location, Y/Latitude Location

The x and y coordinates of the object's location.

Layer Name

The layer name to which the display object is assigned.

Layer Shown, Selectable in Edit Mode, Low Zoom Level, High Zoom Level

These fields only display information about the layer to which the display object belongs. Therefore, the values for these fields can only be modified through the [Screen Layer case information dialog](#) or the [Screen Layer dialog](#).

Anchored

This field indicates whether or not the display object is an [anchored object](#). If the display object can not be an anchored object, then the field is empty.

Font Size

For text objects, this field shows the font size.

Supplemental Data

Supplemental Data can be used to identify display objects that do not represent power system elements. User defined data types can be created and linked to display objects for use in filtering, dynamic formatting, Select by Criteria, and custom hints.

Supplemental Classification Definition

A Supplemental Classification is used to group Supplemental Data records into categories. As an example, if using Supplemental Data records to identify geographical borders, a Supplemental Classification could be *Country*. Each Supplemental Data record uses exactly one Supplemental Classification record.

Supplemental Classification records are defined by one key field:

Name

This field uniquely identifies the classification and cannot be blank.

Supplemental Data Definition

Supplemental data records are defined by two required key fields:

Classification

This is the Supplemental Classification Name. This field cannot be blank.

Name

This is a unique identifier used to further define the Classification. If using supplemental data records to identify geographical borders, the Name could be *United States* to go with a Classification of *Country*. This field cannot be blank.

In addition to the Classification and Name, geographic information, expressions, and custom strings, integers, and floating point values can be assigned to supplemental data records. This information can be entered in the Supplemental Data Case Information Display discussed below.

Displaying Supplemental Data Records

Supplemental Data records are displayed in the Supplemental Data Case Information Display found on the [Model Explorer](#) under **Case Information and Auxiliary > Supplemental Data**. This display consists of two tabs, **Supplemental Classification** and **Supplemental Data**. The table on the Supplemental Classification tab lists all of the Classifications that have been defined. The table on the Supplemental Data tab lists all of the Supplemental Data records that have been defined and provides the means for creating new Supplemental Data records AND Supplemental Classification records. Supplemental Classification records cannot be created under the Supplemental Classification tab.

Creating Supplemental Data Records

To create a new Supplemental Data record, from the **Supplemental Data** tab, right-click on the table and choose **Insert** from the local menu. The **Define Supplemental Data** dialog will open.



Either select an existing **Classification** from the drop-down box or create a new Classification by clicking **New**. (If creating a new Classification, another dialog box will open with a prompt for the Classification Name.) Type in a **Name** for the record. To create the record and close the dialog click **OK**. To create the record without closing the dialog click **Save**. To exit the dialog without creating the record click **Cancel**.

Linking Display Objects to Supplemental Data

Display objects that cannot be linked to a power system element can be linked to a Supplemental Data record. These display objects include Background Line, Background Rectangle, Background Ellipse, Background Picture, Text, Case Information Memo, Online Field, Online Link, Document Link, Memo Text, and Generic Model Field.

Linking any of these objects to supplemental data must be done through the **Display Explorer** found under **Onelines > All Display Objects**. From the **Explore** tab, choose the display object type to link. If they are not already displayed, the **Supplemental Data Classification** and **Supplemental Data Name** fields must be added to the list of fields in the table. Simply enter the appropriate Classification and Name in the fields to link a particular display object to a Supplemental Data record.

Once a display object is linked to a supplemental data record, the information in the supplemental data record can be used to filter, dynamically format, Select by Criteria, and display custom hints for that display object.

Because there is no requirement to link display objects that cannot be linked to power system elements to supplemental data records, these objects will not be highlighted along with other unlinked elements when they are not linked to a supplemental data record.

Automatic Linking of Supplemental Data When Inserting Geographic Information

Background objects (lines and ellipses) created from inserting objects from borders included with Simulator and shapefiles can be linked automatically to supplemental data records.

When [auto inserting borders](#) included with Simulator, the option to **Automatically link county and state borders to supplemental data** needs to be checked for this to occur.

When inserting background objects from shapefiles, there are several options that need to be set for proper linking to supplemental data. These options are available on the **Identify** tab of the [GIS Shapefile Data Dialog](#).

Movie Maker

Simulator no longer contains a built-in movie maker tool. Instead, PowerWorld Corporation recommends other movie making tools that can be used from outside of Simulator. Please refer to the PowerWorld website at www.powerworld.com/support/videocapture.asp for references to these third party movie tools.

Map Projections

Background on Map Projections

In order to create a map of a spherical object such as the Earth it is first necessary to convert the Earth coordinates of latitude and longitude into a two-dimensional coordinate system needed for a map. This conversion from the spherical coordinates of latitude and longitude to some xy Cartesian coordinate system is called a *map projection*. Mapmakers and mathematicians throughout history have invented many map projections. For each map, the mapmaker must decide which projection will look best for his or her purposes. This will depend on what part of the globe is being drawn, the size of the area being drawn, and the shape of the object being drawn. In general, map projections fall into three standard classes: cylindrical, conical, and planar. For a complete analysis of the topic of map projections see reference 1.

For maps of North America, PowerWorld supports a Simple Conic map projection which is described below.

For maps of the entire Earth, PowerWorld supports a Mercator projection which is described below.

Simple Equidistant Conic Map Projection

For maps made of the 48 conterminous states in the United States, maps are often made using one of the conical map projections. These projections can be recognized by the constant latitude lines that are seen as arcs. When looking at the U.S./Canadian border in the western United States, you will see that it curves on maps made using a conical projection, even though this is a constant latitude border. Several conical projections exist including the Albers Equal-Area Projection, Lambert Conformal Projection, and Simple Equidistant Projection. They can be visualized as taking the globe and placing a cone over it as shown in Figure 1 and then using this to map the coordinates to the xy plane shown at the right. Constant longitude lines are seen as straight lines and constant latitude lines are seen as arcs on the resulting map.

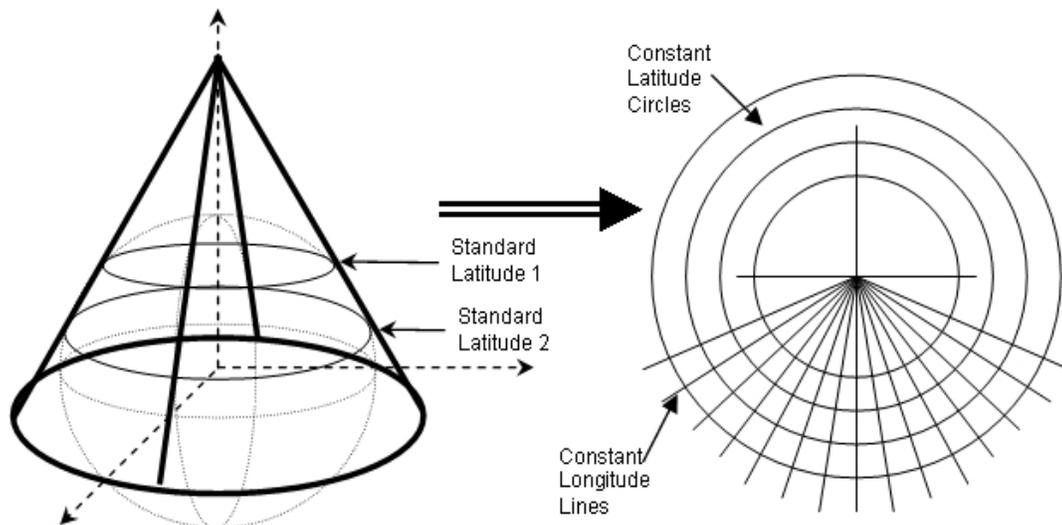


Figure 1 Conic Map Projection

The equations for converting back and forth between (longitude,latitude) and (x,y) using the simple equidistant projection are described next. These equations are taken from reference 1.

First, you must specify seven values that are parameters of the mapping:

- R = Radius of the sphere (earth) in the units used on the map
 ϕ_o = The latitude of the "shifted" origin
 λ_o = The longitude of the "shifted" origin
 ϕ_1 = The first standard parallel
 ϕ_2 = The second standard parallel
 x_f = The shift in the x-direction. Referred to as "false-easting" in some map projections. This is done so that all coordinates are positive.
 y_f = The shift in the y-direction. Referred to as "false-northing" in some map projections. This is done so that all coordinates are positive.

For these values, the following variables are defined:

$$\rho_o = R(G - \phi_o) \quad G = \frac{\cos \phi_1}{n} + \phi_1 \quad n = \frac{\cos \phi_1 - \cos \phi_2}{\phi_2 - \phi_1}$$

Then given a latitude = ϕ and a longitude = λ , calculate the x and y coordinate using

$$\begin{aligned} \rho &= R(G - \phi) \\ \theta &= n(\lambda - \lambda_o) \\ x &= \rho \sin \theta + x_f \\ y &= \rho_o - \rho \cos \theta + y_f \end{aligned}$$

And given an x and y coordinate, calculate the latitude(ϕ) and longitude(λ) using

$$\begin{aligned} \rho &= \pm \sqrt{(x - x_f)^2 + (\rho_o - y + y_f)^2} \\ &\text{taking the sign of } n \\ \theta &= \arctan \left[\frac{x - x_f}{\rho_o - y + y_f} \right] \\ \phi &= G - \frac{\rho}{R} \\ \lambda &= \lambda_o + \frac{\theta}{n} \end{aligned}$$

Figure 2 shows more clearly what some of these variables are.

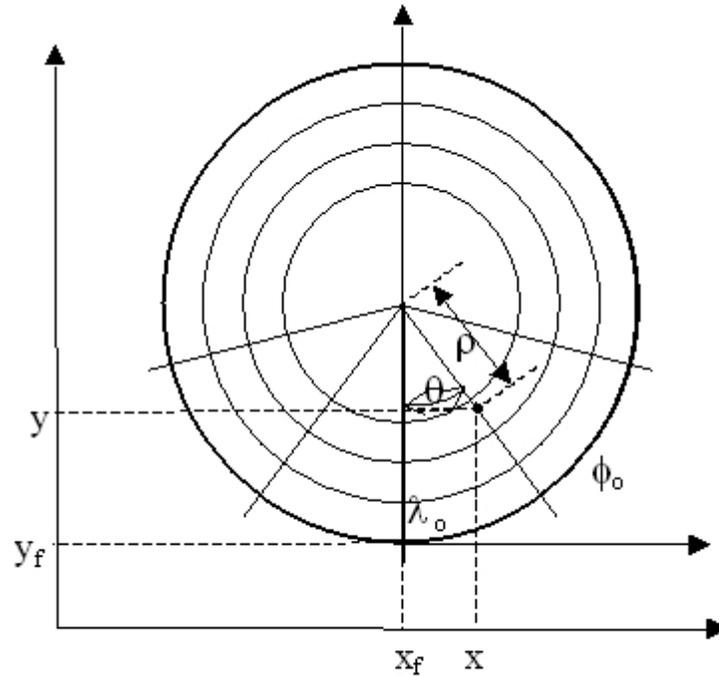


Figure 2 Defining Variables for Conic Projections

Maps made using PowerWorld's built-in map borders for North America use the following parameters:

$$\begin{aligned}
 R &= 30700.0 \\
 \phi_0 &= +9.7 \text{ degrees} \\
 \lambda_0 &= -96.5 \text{ degrees (negative because in Western Hemisphere)} \\
 \phi_1 &= 0.00 \text{ degrees} \\
 \phi_2 &= 50.00 \text{ degrees} \\
 X_f &= 20968 \\
 Y_f &= 0
 \end{aligned}$$

These numbers were thus set to be the default map projection numbers within PowerWorld. This projection will work well for the one-line diagrams with PowerWorld Corporation has created on the same standard background.

Mercator Projection Use in PowerWorld

For the Mercator Projection, PowerWorld assumes $R = 30700.0$ units. From this, given a latitude $= \phi$ and a longitude $= \lambda$, calculate the x and y coordinate using

$$\begin{aligned}
 x &= R * \lambda \\
 y &= R * \ln \left[\tan \left(\frac{\pi}{4} + \frac{\phi}{2} \right) \right]
 \end{aligned}$$

And given an x and y coordinate, we calculate the latitude(ϕ) and longitude(λ) using

$$\lambda = \frac{x}{R}$$

$$\phi = \tan^{-1} \left[\sinh \left(\frac{y}{R} \right) \right]$$

References

1. Snyder, John P., Map Projections - A Working Manual, U. Geological Survey Professional Paper 1395, United States Government Printing Office, Washington, D.C.: 1987.

Great Circle Distance Dialog

To find the Great Circle Distance between any two points of longitude, latitude, select **GIS Tools > Great Circle Distance** from the **Active** ribbon group on the **Onelines** ribbon tab and the Great Circle Distance Dialog will be displayed. This can be done in Edit or Run Mode.

The Great Circle Distance Dialog allows the calculation of distance between two points of longitude, latitude. The [Great Circle Distance Calculation](#) assumes that the earth is a sphere and makes no adjustments for the actual flattening of the earth. The two longitude, latitude points can either be entered manually for any two valid points or can be chosen from valid longitude, latitude values stored with either buses or substations. Valid longitude values are between -180 and 180 degrees. Valid latitude values are between -90 and 90 degrees. The calculated distance is given in either kilometers or miles.

From Location

Click the **Choose From Location** button to choose either the location of a bus or substation as the **From Location**. The **Latitude (degrees)** and **Longitude (degrees)** entries for the From Location will be filled in with the latitude, longitude information that is stored with the chosen element in the case data. These entries are not based on the location of a display object, if one exists, of the chosen element. If the element does not have valid geographic information stored, the values that are entered for the latitude, longitude will not be valid.

The Latitude, Longitude entries can also be entered manually.

To Location

Click the **Choose To Location** button to choose either the location of a bus or substation as the **To Location**. The **Latitude (degrees)** and **Longitude (degrees)** entries for the To Location will be filled in with the latitude, longitude information that is stored with the chosen element in the case data. These entries are not based on the location of a display object, if one exists, of the chosen element. If the element does not have valid geographic information stored, the values that are entered for the latitude, longitude will not be valid.

The Latitude, Longitude entries can also be entered manually.

Units

Choose the units in which to report the calculated distance.

Calculate Distance

Click this button to calculate the distance in the chosen Units between the From Location and the To Location. The calculated distance will appear in the box above the button. *Error in input data* will be returned if there is a problem with any of the latitude, longitude information.

Great Circle Distance Calculation

The Great Circle Distance Calculation allows the calculation of distance between two points of longitude, latitude. The Great Circle Distance Calculation assumes that the earth is a sphere and makes no adjustments for the actual flattening of the earth.

The calculation consists of the following parameters:

- R = Radius of the Earth (6378.137 km from NAD83/WGS84 datum)
- ϕ_0 = The latitude (radians) of the starting location
- λ_0 = The longitude (radians) of the starting location
- ϕ_1 = The latitude (radians) of the ending location
- λ_1 = The longitude (radians) of the ending location
- d = Distance in kilometers between starting and ending location

The distance in kilometers between the starting and ending location is determined from the following equation:

$$d = 2R \arcsin \left(\sqrt{\sin^2 \left(\frac{\phi_0 - \phi_1}{2} \right) + \cos(\phi_0) \cos(\phi_1) \sin^2 \left(\frac{\lambda_0 - \lambda_1}{2} \right)} \right)$$

This calculation is used with several GIS tools to determine the distance between two locations or facilities on a diagram.

Populate Lon,Lat with Display X,Y

To populate the longitude,latitude fields of buses and substations with the locations of corresponding display objects converted to longitude,latitude using the selected projection, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Populate Lon,Lat with Display X,Y**.

The Populate Lon,Lat with Display X,Y Dialog will be displayed. This can be done in either Edit or Run Mode. The dialog will show which map projection is currently in use for the oneline. Users should be careful to not change the map projection once one has been established so that all objects on the oneline are consistently using the same projection. If no projection has been set, the dialog shows **None (x,y)** as the Map Projection, selecting a map projection will establish that map projection as the current oneline map projection.

Map Projection

None (x,y)

No map projection is in use.

North American (simple conic)

This projection is better when dealing with longitude, latitude points in North America. This is the projection that has been used in Simulator for years in defining geographic borders and placing them on a oneline.

Entire World (Mercator)

This projection is better when dealing with longitude, latitude points spread throughout the world.

Click **Populate Longitude,Latitude** to complete the conversion. The longitude, latitude values converted from display x, y coordinates can be viewed on the bus and substation case information displays or the information dialogs.

Geography/Coordinates

The option is available to display screen coordinates in longitude, latitude instead of x,y. This option can be set by selecting **Online Display Options** from the [Onelines](#) ribbon tab or the [Options](#) ribbon tab. The following options are located under the **Geography/Coordinates** category.

[Map Projection](#) in Use

The map projection indicates which projection has been selected for inserting objects on the online. Users should be careful to not change a projection once one has been established. Otherwise, there is the risk that different objects will be inserted with different projections.

Show longitude, latitude coordinates when showing x, y coordinates

When this option is checked and a map projection other than *None (x,y)* is selected, longitude, latitude coordinates will be displayed anywhere that x,y screen coordinates are normally displayed. This option will only be enabled when a map projection other than *None (x,y)* has been selected.

Shape File Import

Simulator allows you to import ESRI shapefiles (*.shp/*.dbf pairs) as a group of background lines or points on a oneline diagram. There are several locations where you may open the GIS Shapefile Import Dialog.

- Go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Insert GIS Data from Shapefile**.
- Go to the [Draw](#) ribbon tab, then on the [Individual Insert](#) ribbon group choose **Background > Insert GIS Data from Shapefile**
- Go to the [Draw](#) ribbon tab, then on the [Quick Insert](#) ribbon group choose **Auto Insert > Insert GIS Data from Shapefile**

The GIS Shapefile Data Dialog has the following sections:

[Control](#)

[Modify](#)

[Identify](#)

[Format](#)

[Shape List](#)

GIS Shapefile Data: Control

This is the first page of the [GIS Shapefile Data](#) dialog. The control page contains the controls for reading a shapefile into Simulator and provides some basic information on the shapes read from the file.

Read in Shapefile

Pressing this button will allow you to choose a shapefile and will load it into Simulator for placement on the currently selected online diagram.

Save Shapefile

Clicking this button will save to file the shapefile information currently loaded in memory. This option is useful if making changes to shape format attributes or eliminating certain shapes from those read in and then saving these changes in the shapefile format.

Shapefile Information

Once the shapefile has been loaded into memory, this section will be populated with general information on the number of shapes loaded, and the maximum and minimum X and Y values (usually in longitude and latitude.)

Transfer Shapefile Objects to Online

If you are ready to place the shapes on the online diagram, but want to continue working with existing shapes or read more shapes from another file, press the **Transfer Shapes to Online and Clear** button. This will keep the dialog open following the transfer. It also gives you options of what should happen to the current shapes in memory after you have transferred them to the diagram. You can choose to clear only the shapes that were transferred, all of the shapes, or none of the shapes. The first choice relates to options you have on the [Shapefile Objects](#) page regarding whether or not to transfer certain shapes from the shapefile to the diagram.

Insert Into Layer

Before transferring the shapes to the online diagram, you can choose which [layer](#) the shapes should be assigned to. If you wish to place the shapes into a new undefined layer, click on the **Define Layer** button to access the [Screen Layers](#) table.

Transfer Shapes to Online and Close

If you wish to transfer the currently loaded shapes to the online and close the dialog (clearing the shapes from memory), press this button to complete the process.

GIS Shapefile Data: Identify

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, identification options can be specified on the Identify page.

Identification String for Display Auxiliary Files

Shapefile shapes are created as background objects when added to a diagram. If using [display auxiliary files](#) to create or edit diagrams, objects must be uniquely identified. Because background objects typically do not link to an object in the power flow case, and therefore have no other unique means of identification, a unique ID string must be assigned to objects created via shapefiles. This unique identifier populates the **Auxiliary ID** field used in [display auxiliary files](#) and can be viewed from the [Display Explorer](#).

Prefix

Optionally, a prefix can be concatenated with the selected Attribute to further identify the shape.

Attribute

The drop-down box is populated with the fields that are available with the .dbf file. Choose a field whose value will be used to uniquely identify the shape. By default, the *Record Number* of the shape is used for identification if no other Attribute is selected.

Suffix

Optionally, a suffix can be concatenated with the selected Attribute to further identify the shape.

Linkage to Supplemental Data

Background objects created from shapefiles can be linked to [Supplemental Data](#) records for purposes of further identifying the objects. This is useful when applying filters, applying dynamic formatting, using Select By Criteria, or defining custom hints with display objects created from shapefiles.

Link Display Objects to Supplemental Data

Check this box to link the background objects created from shapefile shapes to Supplemental Data records.

Supplemental Classification

Select an existing Supplemental Classification from the drop-down box or click **New** to create a new Supplemental Classification.

Attribute for Supplemental Data Name

Select an Attribute from the drop-down box. The list of attributes is populated from the fields that are available in the .dbf file. For each unique Attribute value, a supplemental data record will be created with the chosen Supplemental Classification and with its Name set to the Attribute value.

GIS Shapefile Data: Modify

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Modify page can be used to modify the XY attributes of the shapes.

Map Projection

There are two map projections you can choose from when placing the shapes onto a online diagram. The choices are:

- North American (simple conic)
- Entire World (Mercator)

Once you have selected which projection to use, click on the **Convert to Specified Map Projection** button to process the conversion on the shapes currently in memory.

Shift/Scale Shapefile X/Y Data

You also have the option to shift the XY coordinates of shapes or scale them by scalar values. To shift shapes, enter scalar shift values for the X value and Y value and press the **Shift XY Data** button. To scale the XY coordinates, enter scalar scaling factors for the X and Y value, and press the **Scale XY Data** button.

GIS Shapefile Data: Format

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Format page can be used to modify the appearance attributes of the shapes.

Change Shape Format Characteristics

Point Size

The size of points read from the shapefile can be adjusted when transferred to the online diagram.

Line/Border Thickness

Choose the pixel thickness of the shapefile lines when transferred to the online diagram.

Line/Border Color

Set the line color of the shapefile lines when transferred to the online diagram.

Fill Color for Points/Polygons

Set the fill color of the shapefile points and polygons when transferred to the online diagram. The box labeled **Use Fill Color** must be checked as well.

Stack Level

Indicate which [stack level](#) should be applied to the shapes when they are transferred to the online diagram.

Immobile

Check this box if you wish for the shape file objects to be immobile once they are transferred to the online diagram. This will prevent you from inadvertently selecting and moving the shapefile objects on the online diagram while in [Edit mode](#).

Set all shape format attributes to the values above

Once you have finished changing the format settings for the shapes, press this button to apply them for the transfer of shapes to the online diagram.

Automatic Color Mapping

Available Attributes

To customize the shapefile shapes based on a field value read from the .dbf file, select an attribute from the list.

Color Map

Select a color map to apply to the selected attribute. To modify existing or create new color maps, click on the **Modify Color Maps** button. The color of the object will then be based on where its selected attribute value maps to the selected color map.

Change Border Color, Change Fill Color

These two boxes allow you to choose if the color map selected should apply to the border color, the fill color, or both.

Change Color Based on Attribute/Color Map

Once you have finished setting the color map for a selected attribute, press this button to apply the settings before the transfer of shapes to the online diagram.

GIS Shapefile Data: Shape List

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Shape List page can be used to flag shapes for inclusion in the transfer and customize the appearance attributes of specific shapes.

This page will be blank until you have actually read a shapefile into memory. Once you have done so, you will see a list of all shapes read from the file. The table allows you to customize the appearance and attributes of a specific shape, similar to the options on the [Format](#) page.

In addition, you also can choose to include or exclude shapes from the transfer to the online diagram. This gives you the greatest flexibility for hand-picking which shapes you want to add to a online diagram.

Shapefile Database Record Dialog

The Shapefile Database Record dialog is available by selecting **Show Shapefile Fields...** from the local popup menu (obtained by right-clicking on a display object) of any display object that was created from a shapefile. This menu option will not be available for display objects that were not created from a shapefile. This dialog provides information read from the database file (.dbf) associated with the shapefile from which the display object was created. Each time that this dialog is displayed, the database file is read to fill in the field information. The field information is not stored in the PowerWorld display file (.pwd) so the associated database file must be available for this information to be displayed.

File Name

This is the path and name of the database file (.dbf) associated with the shapefile from which the display object was created.

Record Number

This is the record number of the display object within the shapefile.

List of Fields

This list details the fields and values belonging to the given record number in the database file.

Edit Online Browsing Path

Select this option to update the Online Browsing Path. This browsing path will be used to search all listed directories for the database file specified in File Name if it cannot be found in the directory specified with the File Name.

Export Oneline As Shapefile

Display objects can be exported into shapefile formats (.shp/.dbf pairs) by going to the [Onelines](#) ribbon tab, then in the [Active](#) ribbon group, choosing **GIS Tools > Export Oneline as Shapefile**. This will open the Select Objects and Fields to Export Dialog, which is similar to the Select by Criteria Dialog.

The dialog is used to select one type of display object to export to the shapefile and the criteria used to select which specific objects will be included in the export.

Selecting Areas and Zones

Use the **Area** and **Zone** fields to select the areas and zones in which to select display objects. [Ranges](#) of area and zone numbers can be entered in the usual way, or the **All Areas** or **All Zone** boxes can be checked to select all areas and all zones respectively. The **Areas** and **Zones** tab pages can also be used to select the areas and zones in which to select display objects.

Selecting Voltage Levels

Specify the max and min voltage levels using the boxes that are provided, keeping in mind that all voltages are in kV. The **All Voltages** box can be checked to include all voltages without requiring the specification of a voltage range.

Selecting Layers

All layers can be selected by choosing the **All** option for Layers. If specific layers are to be selected, the **Range** option should be selected and the **Layers** tab should be used to check and uncheck specific layers in which to select display objects.

Type of Drawing Object

Select the type of object to export in the **Type of Drawing Object** dropdown box. Alternatively, the **Find...** button next to the dropdown can be used to select the object type from a list of available object types.

Filter

The **Filter** dropdown box provides a list of filters available for the selected Type of Drawing Object. The **Find...** button next to this dropdown can be used to find a filter or define a new filter for the selected object type. Filter options will not be enabled when an object type that does not allow filtering is selected.

Selecting Fields

The **Fields** list will be populated for object types that have associated fields. The selected fields will be written to the database file (.dbf) along with the value of the field. When the **Specified** option is selected, individual fields can be chosen by checking the box next to specific fields. When the **All** option is selected, then all fields associated with the selected type of object will be included in the database file. When the **Show Only Commonly Used Fields** option is checked, only those fields that are deemed to be commonly used will be displayed in the list. To show all fields associated with the selected object type, uncheck this box. Use the **Check All** and **Uncheck All** buttons to check all or uncheck all of the fields.

Advanced Selection Criteria

If only objects that are currently visible on the display (due to layering, etc.) are to be selected, check the box labeled **Select only currently visible objects**. If some display objects have been selected prior to opening the dialog, an advanced selection option becomes available. If the box

labeled **Use as a filter on presently selected objects** is checked, the criteria chosen in the dialog will only affect the previously selected objects. In other words, only the objects that were previously selected AND that match the chosen criteria will remain selected when exporting to the shapefile.

Export Coordinates in Longitude, Latitude

If a valid map projection has been selected for displaying display object coordinates in longitude, latitude, then the option to **Export coordinates in longitude, latitude** will be enabled. (A valid map projection can be set with the Map Projection in Use Option found in the [Geography/Coordinates](#) category of the Online Display Options dialog.) By default, the Export coordinates in longitude, latitude option will be checked when it is enabled. When checked, the coordinates of the display objects will be exported to the shapefile in longitude, latitude instead of PowerWorld Simulator x,y coordinates.

Saving Shapefile Export Descriptions

Shapefile Export Description settings can be saved with the case by clicking **Save As** to save with a new name or **Save** to save with the current name. This will allow the quick recovery of settings that may have been previously used. Use the **Rename** button to rename an already saved set or the **Delete** button to remove a previously saved set of criteria. If Shapefile Export Description sets have been saved, they can be exported to an auxiliary file by choosing **Save to AUX file**. The **Load from AUX file** button will load an entire auxiliary file regardless of whether it contains Shapefile Export Descriptions or not.

Saving the Shapefile

After all criteria have been set, click **OK**. A dialog will open prompting for the name of the file to save. Enter a file name and click **Save**. The type of file that is specified in the dialog is a shapefile with .shp extension, however, three files will be saved. All files will have the name specified in the save dialog, but the extensions will be .shp, .dbf, and .shx.

Insert Measure Line

Distances between display objects can be approximated by inserting a measure line on the display. A measure line is similar to a background line. It can be drawn as a straight line between two points or as a line with many vertices and segments. By default, a measure line is drawn as a yellow line with a pixel thickness of 4.

To insert a measure line go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Insert Measure Line**. Then position the mouse cursor on the display where the line should begin and click and release the left mouse button. Move the mouse to the desired termination point of the first line segment. A straight segment will follow the mouse movements. Click and release the left mouse button to complete the line segment and to prepare for drawing the next segment, or double-click if this is the last line segment. To draw a freehand shape rather than a series of straight line segments, click and hold the left mouse button where the freehand shape begins and drag the mouse to trace the desired shape (while holding the left mouse button down). Release the left mouse button to complete the section of the freehand shape. At this point, either another freehand section or a straight line segment can be added. When all desired freehand and straight line segments have been added, double-click the left mouse button to complete the line.

To change the shape of the line, first left click on the line to select it. Handles will appear at each vertex. A vertex can be moved by holding the left mouse button down and dragging the vertex to a new location. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where the new vertex should be added. Note that freehand lines are nothing more than a continuous series of vertices.

Once the measure line has been added to the display, the length of the line can be displayed by right-clicking on the line and selecting **Measure Length...** from the local menu. The units of the length are dependent on the map projection in use and whether the option to **Show longitude,latitude coordinates when showing x,y coordinates** is checked on the [Oneline Display Options dialog](#). If the map projection in use is something other than x,y and the option to show longitude,latitude coordinates is checked, the length will be given in miles and kilometers. Otherwise, the length will be given in Simulator units. The [Great Circle Distance calculation](#) is used to determine the distance between vertices on the line to determine the approximate length in miles and kilometers.

Delete All Measure Lines

All [measure lines](#) can be deleted from the display. To do this go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Delete All Measure Lines**.

Path Distances from Bus or Group

To access the Determine Path Distances from Bus or Group dialog go to the [Tools](#) ribbon tab, and choose **Connections > Determine Path Distances to Bus** from the [Other Tools](#) ribbon group. The Determine Path Distances from Bus or Group dialog provides a way to calculate how far electrically every bus in the system is from a particular starting point.

After clicking the **Calculate** button, all buses that are part of the start element will be labeled internally as being a path distance zero. Then a calculation will be done that determines for *each bus* in the system the shortest total path length in getting from each bus to the Start Element. Buses that cannot reach the Start Element will be flagged with a very large distance (by default 10,000). The results of the calculation will then be copied into the Bus Field specified under the Bus Field to Populate option. The results will be displayed in a case information display at the bottom of the dialog. The options on the dialog are described below.

Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X - Per unit series reactance.

|Z| - Magnitude of the series impedance (based on the per unit reactance and resistance).

Length - Length field for each branch.

Number of Nodes - Length of 1.0 is used for all branches.

Other - When choosing Other, click the **Find..** button to choose any numeric field of a branch. Lines to Process

Lines to Process

Regardless of the Distance Measure above, you can choose which branches allowed to be traversed when finding the shortest path.

All - All branches are allowed to be traversed.

Only Closed - Only branches that are presently closed can be traversed.

Filter - Only branches that meet the advanced filter specified can be traversed. Click **Define...** to choose or create an advanced filter.

Selected - Only branches whose Selected? field is set to *YES* can be traversed. Bus Field to Populate

Bus Field to Populate

Choose a field into which the resulting path distances will be copied. It is recommended that you use the custom floating point or integer fields for this purpose.

Closest Facilities to Point

To produce a list of display objects and their distance from a point selected on the one-line diagram, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Closest Facilities to Point**. After selecting this option, the mouse cursor will become a crosshair. Move the cursor to a location on the diagram and left-click. A dialog will open listing all of the display objects and their distance from the selected point in ascending distance from the point.

Select to Show Display Objects of Type

This allows the option of showing all display objects in one list or only listing objects of a particular type.

Distance

This field is listed by default with the display objects. This shows the distance between a given display object and the selected point. The location of the selected point is given by the **X/Longitude** and **Y/Latitude** fields on the dialog. The location of a given object is provided by the **X/Longitude Location** and **Y/Latitude Location** fields with the display object. The units of the distance is given in the Units field on the dialog.

X/Longitude and Y/Latitude

Each display object listed has a corresponding **X/Longitude Location** and **Y/Latitude Location** field. The **X/Longitude** and **Y/Latitude** fields given on the dialog provide the location of the point that was selected on the diagram. The fields on the dialog can also be manually adjusted to determine the distance between the display objects and a given coordinate.

All of the fields that give the location of either the display object or the point selected on the diagram are given in x,y coordinates if a valid map projection is not selected or the option to show coordinates in longitude, latitude is not selected. Otherwise, the coordinates will be longitude, latitude. Both of these options can be set from the Oneline Display Options dialog in the [Geography/Coordinates](#) category.

Units

This indicates the units that the distance is in. The units will be in PowerWorld units if a valid map projection is not selected or the option to show coordinates in longitude, latitude is not selected. Otherwise, the units will be in either miles or kilometers. To display the distance in miles, the option to use the **English Measurement System** must be selected on the PowerWorld Simulator Options dialog under the [Environment](#) category. Select **Metric (SI)** as the Measurement System to display the distance in kilometers.

Recalculate Distances

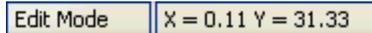
Click this button to recalculate distances after changing the coordinates of the selected point by entering new values for the X/Longitude or Y/Latitude fields.

Oneline Screen Coordinates

Onelines can be any size and can contain any number of objects. The size and position of objects on the screen are specified in terms of x-y "oneline screen coordinates." When the zoom level is 100% the size of the oneline in screen coordinates is 100 by 100. More generally, the size of the oneline is 100^2 divided by the zoom level in both the x and y directions.

The default screen center is the point (50,50) but this can be easily changed (See [Oneline Panning and Zooming](#).) Negative screen coordinates are allowed. Usually you will not have to be concerned about an object's location in screen coordinates.

The [Status Bar](#) displays the current cursor screen coordinates while in Edit Mode.

A screenshot of a status bar from a software application. It consists of two adjacent rectangular boxes. The left box has a thin blue border and contains the text "Edit Mode". The right box has a thin grey border and contains the text "X = 0.11 Y = 31.33".

Status Bar Showing Screen Coordinates

Online Zooming and Panning

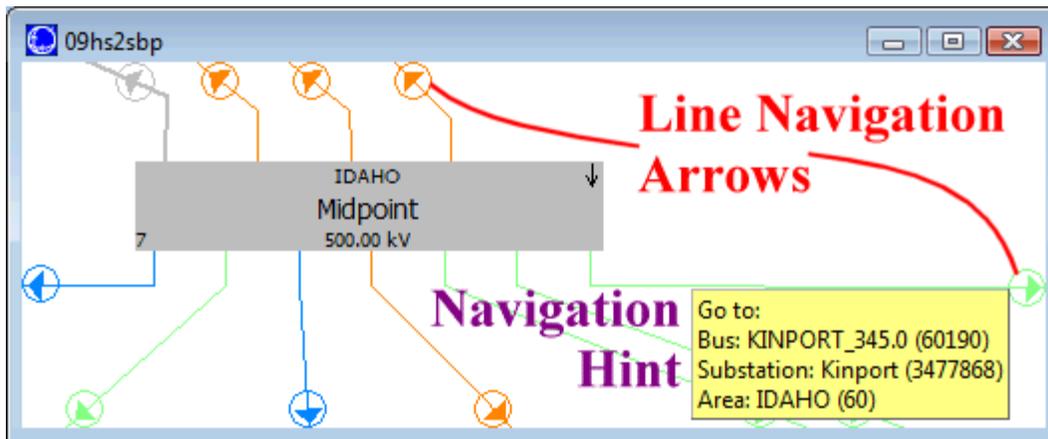
All oneline diagrams permit zooming and panning. Zooming and panning are very important tools for building and investigating large oneline diagrams that occupy more than a single screen of viewing area. Moreover, the display of various objects can be set to depend on the zoom level. See [Online Conditional Display of Objects](#) for more information on this feature.

The following mechanisms are provided for zooming or panning a oneline:

Line Navigation Arrows

For transmission lines whose end-points are not visible at the present pan/zoom level, small Line Navigation Arrows will appear where the line exits the present window. If you click on these arrows, the oneline will automatically pan over to the other end of the transmission line. The oneline will zoom out as it moves away from the present position and zoom back in until it finishes at the same zoom level with the endpoint in the middle of the oneline. On the [Online Tab](#) of the [Simulator Options](#) dialog you can control three attributes for these arrows

1. The size in pixels (default is 9)
2. How fast the program pan/zooms to the new location in seconds. (Making this shorter gets you there faster, but you lose some of the overview information of where you're going as you pan to the new location.)
3. Whether to show hints as you move the mouse over the arrows.



Zooming on Onelines

Zooming involves adjusting the oneline diagram's display area by changing the magnification of the view. Zoom in on a oneline to have the screen display less of the complete oneline diagram, and zoom out on a oneline to have the screen display more of the complete oneline diagram.

To zoom on a oneline diagram using the keyboard, follow these instructions:

- Use CTRL-up arrow to zoom in
- Use CTRL-page up to zoom in quickly
- Use CTRL-down arrow to zoom out
- Use CTRL-page down to zoom out quickly
- The [Zoom](#) ribbon group offers additional zooming options:
- Use the **Zoom in on Area** button of the [Zoom](#) ribbon group to select a region on which to zoom.

- Use the **Show Full** button of the [Zoom](#) ribbon group to zoom the display out to show the entire oneline.
- Use the **Find** button of the [Zoom](#) ribbon group to display the [Pan/Zoom Dialog](#).
- Use the **Save View** button of the [Active](#) ribbon group to display the [Save View Dialog](#), or click on the drop down arrow on the Save View button to choose from a list of saved views.
- Use the **Rectangular Zoom Selector** to select a section of the diagram to zoom into using a selection box. After clicking button (symbolized with a magnifying glass over a dashed rectangle,) left click and hold the mouse button down on the diagram, drag the mouse to select part of the diagram inside a box, then release the mouse button. Simulator will zoom into the region selected inside the box.

Panning on onelines

Panning moves the screen's focus point around the oneline diagram. You can pan left, right, up, or down to view different portions of the complete oneline diagram.

To pan around the oneline using the keyboard, follow these instructions:

- Use the arrow keys to move in the desired direction
- Use page up to move up quickly
- Use page down to move down quickly
- Use the Home key to move left quickly
- Use the End key to move right quickly
- Left-click and hold the mouse button down anywhere on the *background* of the diagram. Make sure you do not click on an object on the diagram. While holding the left mouse button down, drag the mouse in any direction to "drag" the diagram in that direction.

You can also use the [Pan/Zoom Dialog](#) to pan to a specific location, or even to a specific bus. You can display the Pan/Zoom Dialog either by pressing the **Find** button on the Zoom ribbon group of the Onelines ribbon tab or by selecting **Pan/Zoom Control** from the oneline diagram's [local menu](#).

Save View Level Dialog

The **Save View Level Dialog** is used to save an (x,y) location and zoom level, contour description, and/or hidden layer application for a oneline diagram in an easily accessible list for quick recall and application to the diagram. This dialog can be called by selecting the **Save/Edit/Delete View** option from the [oneline local menu](#).

You can also define a new layer by going to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **Save View > Save View**.

Recalling a saved view can be done by either right-clicking on the background to access the [oneline local menu](#) and choosing a view from the **Go To View** list, or by clicking on the drop down arrow on the **Save View** button and selecting a view from the resulting list. Selecting a saved view automatically moves the oneline diagram location to the (x,y) coordinates and zoom level, applies a stored contour, and/or applies hidden layers stored with the selected view.

This dialog can be used for creating a new view, editing an existing view, or deleting an existing view. The **Save View Level Dialog** has the following options:

View Name

This is the name that the view will be stored under in the saved view list. This must be a unique name for each view saved with the oneline diagram. By default this field is blank. To edit an existing view, choose the desired view from the drop down list.

Save Display Information in View

When checked, the view will store an x-y coordinate and zoom level to associate with the view. When the view is selected for display, the diagram will center on the x-y coordinate at the defined zoom level.

X-Coordinate

The x-coordinate for the view. This value will default to the current x-coordinate for the oneline diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

Y-Coordinate

The y-coordinate for the view. This value will default to the current y-coordinate for the oneline diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

Zoom Level

The zoom level for the view. This value will default to the current zoom level for the oneline diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

Save Contour Information in View

If a contour is being displayed when the view is created, you can choose to save the [contour](#) information with the view by checking this box. Saving a contour with a view will display that contour when you switch to that view. You also have the option of saving a blank contour with the view. This is indicated by a Contour Object of 'None' and a Contour Field of 'None.' If you save a blank contour, no contour will be displayed when switching to that view. If you don't save any contour with the view, then any existing contour will be applied when switching to that view.

Contour Object

This section displays the type of object the stored contour pertains to. This field cannot be changed from this location.

Contour Field

Displays the type of value the stored contour pertains to. This field cannot be changed from this location.

Link to Online Display Options Settings

In the [Online Display Options](#) in Simulator, you can save various sets of options with a case for quickly recalling online diagram settings by name. This option allows you to include a custom defined option set by name with the current view. Note that you first must have created and saved a custom set of options for a diagram from the [Online Display Options](#) dialog before this option on the Save Views dialog will be enabled.

Save Hidden Layers

Since Simulator allows the use of [layers](#) to display or hide objects on the diagram, views also can optionally store the layer settings when the view is created. The window below this checkbox will list the layers that are presently hidden on the diagram. When this view is recalled, the layers in this list will be hidden if they were stored with the view.

Save

This button will save a new view or modify the values for an existing view of the name in the **View Name** field.

Delete

This button will delete the currently open view from the saved view list. The dialog information will default to the saved view information of the previous view in the list.

OK

This button will save a new view or modify the values of an existing view of the name in the **View Name** field, and will close the **Save View Level Dialog**.

Save to AXD

By clicking this button, all the views will be saved to a [display auxiliary file](#).

Load from AXD

This button will load all the views saved in a [display auxiliary file](#).

Window Ribbon Tab

The Window ribbon consists of buttons and menus that provide access to customizing the windows in the user interface. It also provides access to help topics and topics regarding the Auxiliary File Format. The ribbon is shown below.



The Case Information ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

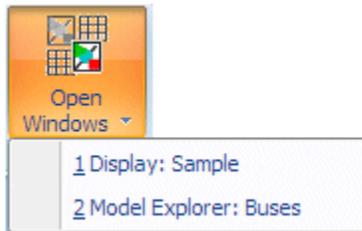
Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Window Ribbon Group

Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



Switch to Free-Floating Windows (Switch to Standard Container Window)

Choosing this option toggles the program from between two different window environment

1. All diagrams and dialogs are contained within the Simulator program shell
2. All diagrams and dialogs are free-floating on your monitor.

For more information see the [Window Basics](#).

Reset to Defaults

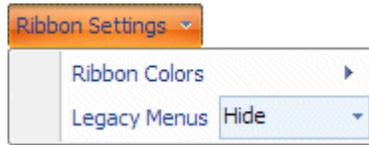
The program keeps track of dialog Free-Floating/Contained Style, Size, and Position. Clicking this button will reset all these to the factory default settings. Nearly all dialogs in will then be reset back to Contained with a default size and position as well. For more information on Style, Size and Position, see [Window Basics](#).

Refresh Displays

Redraws (refreshes) each of the open windows. Simulator usually automatically refreshes the open windows as necessary. However, this action allows you to trigger the refresh when you want it.

Ribbon Settings Menu

This menu contains options for customizing your ribbon.



Ribbon Colors : You can change the color scheme to either **Blue**, **Silver** or **Black**

Legacy Menus : Choose to Hide or Show a special Menus ribbon tab. The Menus ribbon tab mimics the menu and toolbar structure that existed in older versions of Simulator.

Arrange Windows

The options under Arrange Windows are Tile Horizontally, Tile Vertically, and Cascade. Tile rearranges the open oneline diagrams such that the total window area is divided equally among all of them and each is completely visible. Cascade Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

Toggle Full Screen

Toggle full screen makes the currently selected diagram window switch to full screen mode. Full screen mode will dedicate the screen to the window, with all other windows, including the Simulator menus and toolbars, to be hidden. To get back to normal mode, right-click on the diagram in full screen mode and select Toggle Full Screen from the popup menu.

Help Ribbon Group

Contents

Opens the [Simulator Options Dialog](#).

Set Help File

Opens the [Case Description Dialog](#).

About

Click this to open the About dialog. The About dialog contains information about the software as well as showing you the Build Date of your version of Simulator. PowerWorld release patch version of Simulator on our website. These patch versions are named by date and is shown on the About dialog.

There is also an option on the About dialog to Change License Key. This allows selection of a new license key file. Add ons are controlled through the license key. If you are using different versions of Simulator with different add ons, this option provides the mechanism for switching between the different versions of Simulator. You will need a different license key file if you are using different licenses for Simulator that have different add ons.

PowerWorld Website

Click this to open your default web browser to the PowerWorld Corporation Website.

Check for Updates

This button will open the PowerWorld website to the patch download page. You will be required to enter your Account Username and Product ID (password) to check for and download patches for your version of Simulator.

Auxiliary File Ribbon Group

Load Auxiliary

Choose this option to load an [Auxiliary File](#).

Load Display Auxiliary File

Choose this option to load a [Display Auxiliary File](#) to append to the active oneline.

Auxiliary File Format

Opens the [Bus View Oneline](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Export Display Object Fields Menu

Export Case Object Fields Menu

Choose these options to export a list of fields for each type of object in a case (or a Oneline Display). The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.



Oneline Conditional Display of Objects

Along with supporting zooming and panning, the onelines permit the conditional display of objects. That is, it is possible to specify display objects so they are visible only at particular zooming ranges. This enables the oneline to show additional details as the user zooms in and fewer details when the user zooms out.

Please note that the zoom levels are defined as percentages. If you want an object to display only between 50% and 150%, you must select 50 and 150 as the zoom level boundaries.

This option is available by assigning objects on the diagram to a [Layer](#). Layers can be defined and set to objects on the [Format Multiple Objects](#) dialog.

Keyboard Short Cut Actions Dialog

The Keyboard Short Cut Actions dialog allows the user to associate a keyboard shortcut to a determined oneline diagram, and even to a particular [view](#) of that diagram. When the keyboard shortcut is pressed, the oneline diagram is opened (if it was closed) and brought to the front. When a view is also associated, then the user is taken to such a [view](#).

To open the Keyboard Short Cut Actions Dialog, select **Keyboard Shortcuts** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.

The dialog has the following controls:

Keyboard Shortcuts

Lists all the keyboard shortcuts. When selecting a keyboard shortcut, it will show the *oneline* and the *view* associated to the selected keyboard shortcut. If your keyboard has an extra row of function keys, then you can check the box labeled **Include F13-F24**. If a shortcut has been defined for the respective key, then it will be highlighted and a string will summarize the action of the shortcut

Clear All

Clicking this button will delete all the associations of all the listed shortcuts.

Save Shortcut

Click this button after entering a oneline diagram and eventually a view so they can be associated with the selected keyboard shortcut.

Delete Shortcut

Click this button to delete the associations of the selected keyboard shortcut with any oneline diagram and view.

Create Shortcut

This indicates the keyboard shortcut to which the below actions will be assigned. The available actions are to Open a Oneline and to Open an Auxiliary File

Open Oneline

The following options are available for opening oneline actions:

Oneline and View: Oneline

Shows the path and the name of the oneline diagram which will be opened when the selected keyboard shortcut is pressed. If the field is blank, then no oneline diagram is associated to the keyboard shortcut. You can click **Browse** to find the oneline diagram.

Oneline and View: View

Shows the name of the view belonging to the associated oneline diagram to which the user will be taken when the selected keyboard shortcut is pressed. If the field is blank, then no view is associated to the keyboard shortcut.

Action for already open onelines

If the oneline being requested by the shortcut key is already open, then this option determines what should be done. Choices are to close the oneline, bring the oneline to the front, or to open a new oneline.

Where to open oneline

When opening a oneline via a shortcut key, you may do one of the following:

- Separate Window (open the oneline as normal)
- Separate Window – [Toggle Full Screen](#) (open normal and then toggle it to full screen)
- Embed the Active Oneline Window (opens the oneline and embeds it inside the presently active oneline diagram)

Note: if onelines already exist which have been toggled to full screen, and a short cut key is used to open a window that will not be toggled to full screen, then all other onelines present open will be switched to normal mode instead of full screen mode.

Size and Location

This specifies the relative size and location you would like the oneline to be opened at. When opening the oneline normally, the percentage is relative to the size of the container window. When opening the oneline embedded in the active oneline, the percentage is relative to the size of the active oneline.

Make embedded oneline borderless

When [embedding](#) the oneline inside the presently active oneline, this option will make the embedded oneline borderless.

Open Auxiliary File

The following options are available for opening auxiliary file action:

Filename

Shows the path and the name of the auxiliary file which will be opened when the selected keyboard shortcut is pressed. If the field is blank, then no auxiliary file is associated to the keyboard shortcut. You can click **Browse** to find the auxiliary file.

Section name

Shows the name of the [script](#) or [data](#) section belonging to the associated auxiliary file which will be executed when the selected keyboard shortcut is pressed. If the field is blank, then no specific section is associated to the keyboard shortcut, so all script and data sections will be loaded.

Create objects if they do not already exist

Check this box to indicate that new objects in the auxiliary file should automatically be created.

Save/Load shortcuts to/from an Auxiliary File

Click the **Save** button to save all the shortcuts to an auxiliary file. Click the **Load** button to load the shortcuts saved in an auxiliary file. The shortcuts saved in the auxiliary file will replace the shortcuts saved in the dialog.

Close

Closes this dialog.

In addition to the shortcuts displayed on this dialog, Simulator also offers standard windows editing shortcuts such as:

Ctrl + x – Cut Command

Ctrl + c – Copy Command

Ctrl + v – Paste Command

Printing Online Diagrams

To print a online diagram, select **Print Online** from the [Application Button](#) or right-click on the online background and select **Print Window** from the popup menu. This opens the [Print Options Dialog](#), which you can use to configure the print job, including its size, orientation, border, and title bar. You can even choose to print a online to multiple pages using Simulator's multi-page printing.

Print Options Dialog

The Print Options Dialog is used to configure the printing of oneline diagrams (including bus view displays) and of strip charts. To print a oneline diagram, select **Print Oneline** from the [Application Button](#). To print a strip chart, right-click on it to invoke its local menu, and then select **Print Window**. In both cases, Simulator will open the Print Options Dialog.

The Print Options Dialog has three tabs:

Page Layout

This tab sheet contains options for how the diagram will appear on the printed page. It presents the following controls:

Margins

Specify the horizontal and vertical margins in either inches or centimeters.

Scaling

Set this option to **Proportional** to print the oneline diagram or strip chart such that the screen's aspect ratio is maintained. Set this option to **Fit to Page** to force the printout to take up all available space on the page in both horizontal and vertical directions.

Draw Border

Places a border to be drawn around the oneline diagram or strip chart.

The second tab controls the printing of a title bar. Check the **Print Title Bar** box to print a title bar at the bottom of the diagram. Checking this box displays a number of options that allow you to specify various items to include in the title bar. If you don't want to include certain items on the plot, simply leave those fields blank.

The title bar is split horizontally into three sections. If you specify values for all requested items, they will be arranged in the title bar as follows:

Company | Description Line 1 | Date
 Department | Description Line 2 | Drawing #
 Author | Description Line 3 | Title

Title Bar

This tab is used to specify whether or not to include a title bar on the diagram, along with the information that would be contained in the title bar.

Multi-Page

This tab allows the oneline diagram to be "divided" into a specified number of sections for printing the oneline to several pages. A multi-page oneline can be useful for including "close-up" views of areas of a diagram in a report, or the multiple pages can be cut and combined to form a larger printed oneline if a plotter is not available. The **Multi-Page** tab presents the following controls:

Grid Size

Choose this value to subdivide the oneline into an N by N grid. You can then choose which sections of the oneline to print based on the grid overlay. The maximum size is 10 x 10.

Choosing What to Print

A low resolution picture of your oneline is shown on the **Multi-Page** tab. Gridlines are drawn over the diagram to show where the oneline has been subdivided based on the **Grid Size** chosen.

To remove a grid section from the printout, click on the oneline image in the corresponding section of the grid. The selected section of the grid will become dark to indicate you wish to prevent the section from printing.

To add a grid section back into the print job, click on a darkened section of the grid to allow that section of the oneline to be printed.

The number of pages you have chosen to print will be shown on the dialog. When printing out a multi-page oneline with a title bar, the page number will appear on the title bar in the form "Column Letter"-"Row Number". For example, if the Grid Size is 4, the pages are numbered **A-1** through **D-4**.

Click **Print** to send the document to the printer, or **Cancel** to abort the print. Click **Setup** to view the default Windows printer dialog, which will allow you to specify whether to print the figure in portrait or landscape modes and to set various printer-specific properties.

Printer Setup

Choose **Printer Setup** from the [Application Button](#) to configure the printer using the standard Windows printer setup dialog. This dialog allows you to define which printer to use for printing from Simulator, the size of the printed page, the page's orientation, and additional properties that are specific to the printer you are using.

Chapter 3: Creating, Loading, and Saving Simulator Cases

This chapter describes the File and Simulation Control Menus. The File Menu is used to open and save either full cases or oneline diagrams, or to create new cases or new onelines (Edit Mode). The menu is also used to exit Simulator or to validate a case (Edit Mode).

The Simulation Control Menu controls the actual simulation. You can use this menu to do either a time-domain simulation or a single Power Flow Solution.

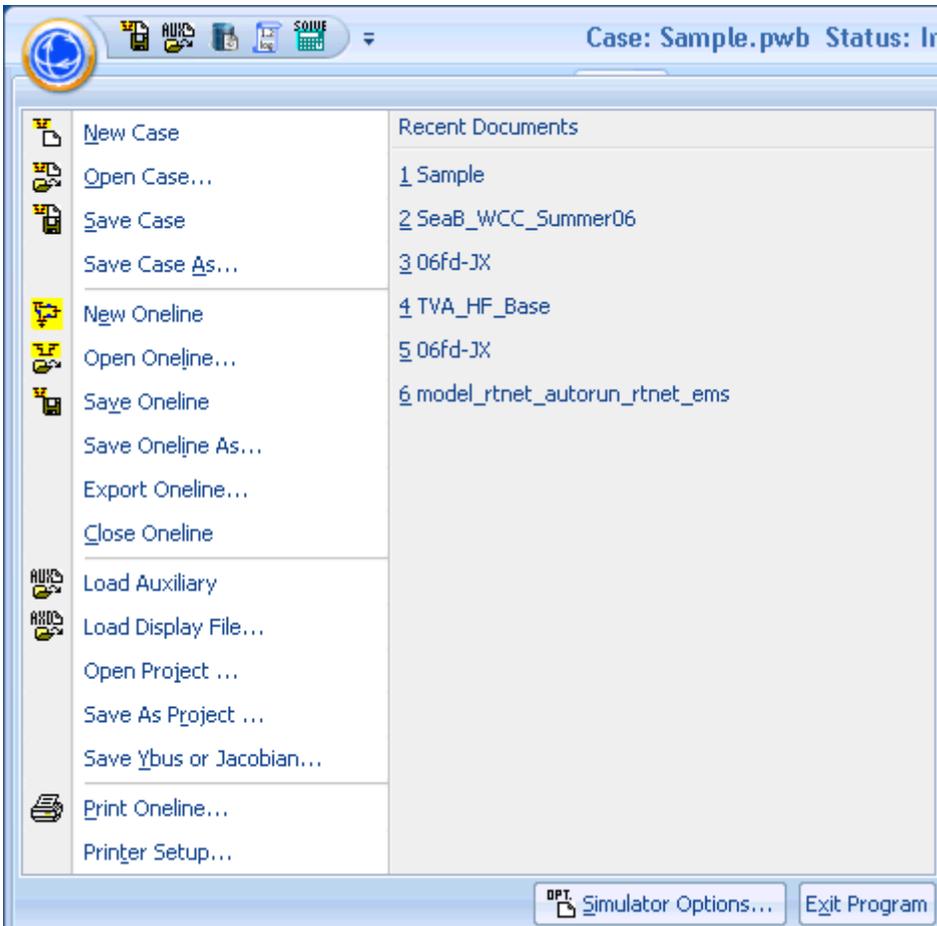
The following material is included:

- File Menu
- Case Formats
- Validating a Case
- Opening Files
- Creating Files
- Closing Files
- Additional File Formats

Application File Menu

The Application File menu is used to open and save either full cases or oneline diagrams, to create new cases, to open and load auxiliary files, to print and to exit Simulator. To access this menu you must click on the Application button in the upper-left corner of the program.

Application Button: 



Case Formats

Simulator supports a number of case formats. You can both open and save cases using most of the following formats:

PowerWorld Binary (*.pwb) (Preferred Format)

For most users, the best choice of power flow case formats is the PowerWorld Binary format. This format stores the most complete set of case information but requires the smallest file sizes. Information stored in this format includes power flow data, economic parameters, case time variation/options values, and screen customizations. The only potential disadvantage of this format is that it is stored in binary form, which means that it cannot be viewed using a standard text editor. All the other formats are ASCII and thus readable in text editors such as Notepad.

One additional benefit of using the Powerworld Binary format is that with this format the case can be password-protected. This setting is available under **Misc. Power Flow > Set Case Username and Password** in the **Case Options** ribbon group on the [Options](#) ribbon tab.

PowerWorld Auxiliary File (*.aux)

Simulator can now save and load cases using the PowerWorld defined auxiliary file format. When the aux file format is chosen as the save file type in Simulator, the data describing the entire case is written to a text file described by Simulator's auxiliary file format. These files can be considerably larger than PowerWorld Binary files, and generally should not be used unless otherwise necessary. Note that not all auxiliary files represent an entire case, and in fact in most cases do not. Only when an auxiliary file is created using the **Save Case As** option in the [Application Button](#) in Simulator does the auxiliary file contain all necessary information for loading the complete case in Simulator as an auxiliary file.

PowerWorld Auxiliary File - Secondary Keys (*.aux)

This option is the same as choosing PowerWorld Auxiliary File *except* for the key identifiers for each record. Instead of using the primary keys (i.e. bus numbers) to identify devices in the data, the secondary keys (i.e. combination of bus name and nominal voltage) will be used as the identifiers in the saved file.

PowerWorld Auxiliary File - Labels (*.aux)

This option is the same as choosing PowerWorld Auxiliary File *except* for the key identifiers for each record. Instead of using the primary keys (i.e. bus numbers) to identify devices in the data, the device primary [labels](#) will be used as the identifiers in the saved file.

PTI Raw Data Format (*.raw)

This format is included primarily for interchange of power flow data with other packages. The PTI Raw Data format only contains power flow data. When using this format, the generator participation factors are automatically set to be proportional to the MVA rating of each generator.

When reading a RAW file that has comments on a given line, the comments will be loaded into the Memo field of the corresponding data record. However, the Memo field will not be written back out when saving a RAW file.

PTI Raw Data Format (*.raw) (with options)

This format is only available when opening a case. This format is the same as the PTI Raw Data Format except that a dialog will open allowing user input on how various options should be set. There is currently only one option to set. This option allows the user to select if transactions should be created for loads that are in different areas than their terminal buses.

Loads may be assigned to a different area than the load's terminal bus. PTI offers an option that allows the user to ignore this assignment when calculating the tie-line flow for an area.

(AREA INT CODE = 1 FOR LINES ONLY)

(AREA INT CODE = 2 FOR LINES AND LOADS)

Simulator does not allow you to define a load this way and then choose to ignore it. Most RAW files that PowerWorld has seen seem to be solved ignoring the loads that are in different areas than their terminal bus by using the "lines only" option. To overcome this, when PowerWorld reads a RAW file, [MW Transactions](#) can be automatically created with the ID "RAW_LOAD". These transactions are between the load's area and its terminal bus and are created so that the export from each area correctly matches a case solved using "lines only". If you know that your case was solved using "lines and loads", then you would not want to add the transactions. There are three options for how to deal with these possible interchange differences. The default is to let Simulator determine what is best based on the input data. If it is known what option was used in PTI, then the user can choose to either create the MW transactions or not create them. See the topic **Add and Remove transactions due to tie-line loads** with [File Management Options](#) for more information about adding or removing these transactions after the case has been loaded.

If not loading the RAW file with options, Simulator will determine based on the input data whether or not the transactions representing loads that are in different areas than their terminal buses need to be created.

When loading the RAW file with options, the user can select how to number the star buses of three-winding transformers. The options are to use a number that is near the primary bus number, start numbering above the maximum bus number, or specify a number at which to start the numbering.

When reading a RAW file that has comments on a given line, the comments will be loaded into the Memo field of the corresponding data record. However, the Memo field will not be written back out when saving a RAW file.

GE PSLF Format (*.epc)

This format is included primarily for importing load flow data saved from GE's PSLF program. The EPC format is a text file format. This format does contain excess information that Simulator does not use, however Simulator does store this information for editing and writing back out to EPC format files.

GE PSLF Format (*.epc) (with options)

This format is the same as the GE PSLF Format except that the [GE EPC File Load Options](#) dialog will be opened to allow user input for various options.

IEEE Common Format (*.cf)

The IEEE Common Format is used to specify only power flow information. CAUTION: IEEE common format does not support many of the formats used in the PowerWorld packages, such as multiple loads and generators at a bus. Usually, IEEE Common Format is used only for inputting cases.

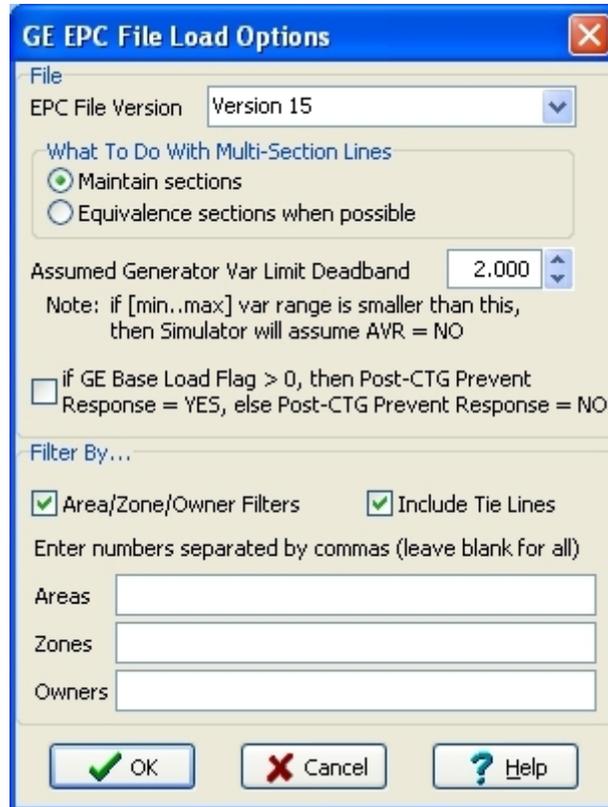
PowerWorld Simulator Project Files (*.pwp)

Project files are described in [Overview of PowerWorld Simulator Project Files](#).

Advanced settings for opening cases are available under the [File Management settings](#).

GE EPC File Load Options

The GE EPC File Load Options dialog will open when choosing to open or save a GE EPC case "with options".



EPC File Version

Set this to determine what version of the EPC file is being read or saved.

What To Do With Multi-Section Lines

Assumed Generator Var Limit Deadband

GE Base Load Flag Setting

When opening a file, the GE Base Load Flag setting will be used to set the Post-CTG Prevent AGC Response field in Simulator. The caption will be the following: **if GE Base Load Flag > 0, then Post-CTG Prevent Response = YES, else Post-CTG Prevent Response = NO**

When saving a file, the Post-CTG Prevent AGC Response field in Simulator will be used to set the Base Load Flag setting in the EPC file. The caption will be the following: **if Post-CTG Prevent Response = YES, then set GE Base Load Flag = 1, else set GE Base Load Flag = 0**

Filter By

Validating a Case

Validation checks for both errors that would prevent a case from working properly in Simulator and for abnormal setup of model objects. Validation errors and warnings are displayed in the [Message Log](#). A sampling of the tasks performed during case validation include:

- Ensures there is a system slack bus assigned
- Ensures the slack bus has an attached generator
- Ensures that no generator is assigned to remotely regulate the slack bus
- Checks for buses having multiple controllable devices attached (i.e. switched shunts)
- Checks all max/min enterable fields to ensure that the max value is greater than the min value

Case validation is performed automatically when switching from Edit Mode to Run Mode. If a case has validation errors, Simulator will not allow the user to switch to Run Mode. Warnings will not prevent switching to Run Mode.

Opening a Simulation Case

The most common first step in using Simulator is to open a case. When a case is opened, any associated oneline files are also opened. To open a case, either select it from the [list of recently opened cases](#) in the file menu or

- Select **Open Case** from the [Application Button](#) to display the Open Dialog.
- In the **Type of Files** box, select the desired file type. By default, the PowerWorld Binary type is selected (*.PWB). The PowerWorld Binary is the preferred file type, providing the most comprehensive power system information along with the smallest size and quickest load time. Other file types include the PowerWorld Case type, PowerWorld Auxiliary file, PTI Raw Data formats, GE EPC text format, and IEEE Common Format. Please see [Case Formats](#) for more details.
- In the list of cases, click on the desired case.
- Click OK.

When opening an existing Simulator case, you may see one or more oneline diagrams pop up. When opening a power flow case created with another program, a oneline may not be available. However, you can easily create a oneline for such a case using the [Edit Mode](#).

If you are in the Edit Mode, you can modify the case. See [Edit Mode Overview](#) for details.

If you are in the Run Mode, the Clock Window may be displayed. This window shows the simulation start time, end time and current time. You may choose to close or open the clock at any time.

If the case has validation errors, the mode is immediately switched to Edit Mode. You must correct the validation errors shown in the message log before you can use the Run Mode to solve the case.

Opening a Online Diagram

Simulator supports online diagrams that have been developed using its Edit Mode tools. PowerWorld can also import PTI Draw Files (*.drw).

When a case is opened, any associated onelines are usually opened, as well. Additionally, online diagrams can be opened directly from any existing onelines using [online links](#). However, you may also directly open a online diagram using the following procedure:

- Select **Open Online** from the [Application Button](#).
- Choose the file type you would like to open (One Line Display File, PTI Draw File, or [Display Auxiliary File](#)).
- In the list of available online files, click on the desired online.
- Click OK.

You may open as many onelines as you like, and even multiple copies of the same online.

The following topics may be helpful when opening onelines created with a different case:

[Refresh Anchors](#)

[Bus Renumbering Dialog](#)

Recently Opened Cases

A numbered list of the most recently opened cases appears in the Recent Files list when you click on the [Application Button](#). Simply choose a case from the list to open the case.

Building a New Case

To create a new case in Simulator, select **New Case** from the [application file menu](#). After asking you whether or not to save the current working case (if one exists), Simulator will automatically switch to Edit Mode. The screen will turn to the default background color, indicating that you can begin to build the new case.

New users may wish to view the tutorial [Creating a New Case from Scratch](#) for further guidance.

Building a New Oneline

This option enables you to create a new oneline diagram. It is accessible from either the Edit or Run modes by choosing **New Oneline** from the [application file menu](#). When you select it, the application will automatically switch to Edit Mode, from which you can construct the new oneline.

New users may wish to view the tutorial [Creating a New Case from Scratch](#) for further guidance. Additionally, the following topics may be of some assistance:

[Using the Insert Palettes](#)

[Oneline Conditional Display of Objects](#)

[Using the Oneline Alignment Grid](#)

Saving Cases

Simulator allows users to save case information in several different formats. To call up the Save As dialog, select **Save As** from the [Application Button](#). The format in which the file will be saved depends upon the value of the **Save as Type** field of the dialog. Simulator can save cases in PowerWorld binary format (the default), PowerWorld Auxiliary file format, PTI raw data, GE PSLF format, and the IEEE common format. Please see [Case Formats](#) for more details on the various case formats available.

To convert a case to a different format, follow this procedure:

- Select **Open Case** from the [Application Button](#).
- In the **Files of Type** box, select the desired file type. By default, the PowerWorld Binary type is selected (*.PWB).
- Select the name of the desired case and click *OK*.
- Select **Save Case As** from the [Application Button](#).
- Change the **Save as Type** setting to match the desired file type, designate the name with which to save the file, and click *OK*.

To simply save the case with its current name and format, select either **Save** from the [Application Button](#) or the **Save Case** button on the Quick Access toolbar.

Saving a Oneline

Select **Save Oneline** from the [Application Button](#) to save the currently selected oneline file.

Select **Save Oneline As** from the [Application Button](#) to save the currently selected oneline file with a different name or format. You may overwrite existing files.

For both the Save Oneline and Save Oneline As options, only the oneline diagram is saved; the case (i.e. the power flow model) is not written to disk.

Close Online

To close a oneline diagram, select to **Close Online** from the [Application Button](#), or right-click on the background of the oneline diagram and select **Form Control > Close** from the local menu. This will prompt you to save the oneline if changes have been made since the last save. Note that this does not close the current case, just the oneline diagram. The power system data is still loaded, and case simulations can be run with the oneline closed.

Auxiliary File Format (*.aux)

Simulator Auxiliary Files have now been greatly enhanced to include both Script sections for running Simulator commands in a "batch" process, and Data sections for changing mass amounts of load flow information. Many [auxilliary file formats](#) used in early versions of Simulator can not be saved in current versions. The older formats can still be read into Simulator.

Auxiliary file format and use now has its own help chapter titled [Auxiliary Script/Data Files](#).

Generator Capability Curves Format (*.gcp)

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The generator capability curves are used to model the dependence of the generator reactive power limits and the generator's real power output. The generator reactive capability curves files have the following format, with one generator per line:

num or name, ID, P₁, Q_{1,max}, Q_{1,min}, P₂, Q_{2,max}, Q_{2,min} ... P_n, Q_{n,max}, Q_{n,min}

where

num/name the generator's bus number or the bus' name in single quotes,

ID the generator's single character id,

P_i a generator MW output value,

Q_{i,max} and Q_{i,min} the associated maximum and minimum reactive power limits.

Version 11 and later of Simulator allow for an unlimited number of capability curve points to be read from the gcp file. Up to 10 different P/Q_{max}/Q_{min} values may be specified in older versions of Simulator.

Generator Cost Data Format (*.gcd)

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The generator cost data file contains the parameters used to model the generator operating costs. Each record in the generator cost data file specifies the operating cost model for a single generator. The cost model may be either a cubic polynomial or a piecewise linear curve. When reading a file of generator cost data records, Simulator will open the [Cost Curve Data Options Dialog](#). This dialog gives you additional control over the input and process of generator cost information.

Each generator cost data record begins with the fields

num or name, ID,

where

num/name the generator's bus number or the bus' name in single quotes,

ID the generator's single character id,

Following the ID field is an optional single-character field **CostModel** that specifies the type of cost model. If **CostModel** is 'CUBIC' or is not specified, the record describes a cubic cost model and thus concludes with the following fields:

a_i, b_i, c_i, d_i, FuelCost ParFac

where

a_i, b_i, c_i, d_i input-output curve coefficients; see [Generator Cost Information](#) for details,

fuelcost fuel cost, expressed in price per Mbtu,

ParFac generator participation factor.

If, on the other hand, **CostModel** is 'PLIN', the record describes a piecewise linear cost model. The remainder of the record specifies pairs of generator output and corresponding generator cost:

FixedCost MW₁ IncCost₁ MW₂ IncCost₂ ... MW_n IncCost_n MWneg ParFac

where

FixedCost Operation cost independent of the generator's MW output; expressed in \$/Hr,

MW_i, IncCost_i Output/incremental cost data point pairs that define the incremental cost model for the generator (note IncCost_i has the units \$/MWhr),

MWneg An arbitrary negative value used to terminate the record,

ParFac generator participation factor.

Generator Cost Curve Data Options

The **Cost Curve Data Options Dialog** gives you additional control over the input and processing of generator cost information. It appears when you try to read generator cost information from an [auxiliary data file](#) (either [.aux](#) or [.gcd](#)).

To command Simulator to turn all units onto automatic generation control (AGC) for which it reads a cost data record, check the first checkbox. This box has the rather lengthy label "If a unit for which a cost curve is read is not on AGC, Simulator should set it on AGC." In other words, if a cost curve is read for a particular generator, that generator will be set on AGC if this box is checked.

Cross-compound is a generator architecture sporting a single boiler and two turbines, one operating at high pressure and the other operating at low pressure. The existence of cross-compound units may complicate the modeling of generator cost characteristics, because sources for this information tend to describe the cost associated with the tandem, whereas Simulator's economic dispatch will try to dispatch the two parts of the cross-compound unit as two separate units. To address this problem, Simulator gives you the option to lump the properties of the high- and low-pressure turbines into a single unit. It consolidates the two components by adding together their maximum MW outputs, minimum MW outputs, and present output levels, assigning these quantities to one of the units, and setting the other unit off AGC and at 0 MW of output. To have Simulator perform this function for you, check the box in the *Cross Compound Units* panel.

To tell Simulator how to identify units that belong to the same cross-compound generator, click the box labeled "How To Identify...". This will open the [How to Identify Cross-Compound Units](#) dialog.

Identify Cross Compound Units

Simulator regards cross-compound units as those that are connected to the same bus and that have generator id's that match any of a number of specified pairs. You must define for Simulator the pairs of generator id's that should be used to identify cross-compound units. Use the "**How to identify cross-compound units**" dialog to do this.

Specify the pair of generator id's in the small text boxes below the text labeled "Specify a new pair of id's that identify two units that belong to the same cross-compound set." Then, click the **Add** button to include the pair of id's you just entered in Simulator's cross-compound identification procedure. The box on the right lists all the pairs of identifiers Simulator will use to identify parts of a cross-compound set. To remove a pair of identifiers from this list, select it from the box and click the Delete key.

The list of cross-compound identifier pairs is stored in the pwrworld.ini file, so there is no need to specify these pairs each time you load in a new case.

Injection Groups Format (*.inj)

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The injection group files store the information for defined injection groups (also referred to as participation groups). The files have the following format:

GROUP *groupname*

POINTS

Devtype busnum id participation pointtype

.

.

.

END

where

groupname the name of the injection group, in single quotes,

devtype either GEN or LOAD,

busnum bus number of the device,

id the device's single character ID,

participation relative amount each device will contribute during a transfer,

pointtype either FIXED or DYNAMIC.

Multiple GROUP sections can be put in the file, each marked with END to signify the end of that group's data.

Interface Data Format (*.inf)

These files are outdated for Simulator version 8 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The interface data files store the interface data. The files have the following format:

INTERFACE intname lima limb limc
Elemtype from to id checktoend fromto

·
·
·

where

intname the interface name, in single quotes,

lima, limb, limc the three limits for the interface, in MW,

elemtype one of three valid element types: AREA, LINE, or ZONE

from for AREA and ZONE, the first area or zone number; for LINE, the from bus number,

to for AREA and ZONE, the second area or zone number; for LINE, the to bus number,

id only included for LINE, the two character circuit identifier; otherwise left blank,

checktoend only included for LINE, if this field equals 1, the flow is checked at the to bus, otherwise the flow is checked at the from bus; otherwise left blank,

fromto only included for LINE, if this field equals 1, the positive flow is assumed to be from the from bus to the to bus, otherwise the flow is assumed to be in the opposite direction; otherwise left blank.

Multiple interface records can be listed in the file, and multiple interface elements can be listed in each interface record.

Sequence Data Format

The sequence data files store the sequence data needed to run a fault analysis. The sequence data format has the following five different types of records:

- Generator
- Load
- Branch
- Switched Shunt
- Mutual Impedance

Each of these four types of sequence data records has a different format, since they each store different information for the sequence data. The format of each record is given below:

Generator

GEN busnum ID Rpos Xpos Rneg Xneg Rzer Xzer RN XN

where

- busnum** generator terminal bus,
- id** generator identifier,
- Rpos, Xpos** positive sequence resistance and reactance,
- Rneg, Xneg** negative sequence resistance and reactance,
- Rzer, Xzer** zero sequence resistance and reactance,
- RN, XN** neutral-to-ground resistance and reactance.

Load

LOAD busnum Gneg Bneg Gzer Bzer

where

- busnum** load terminal bus,
- Gneg, Bneg** negative sequence total conductance and susceptance for all loads at bus,
- Gzer, Bzer** zero sequence total conductance and susceptance for all loads at bus.

Branch

BRANCH fbusnum tbusnum ckt Rzer Xzer Czer fGzer fBzer tGzer tBzer Xftype

where

- fbusnum, tbusnum** from and to bus numbers for the branch,
- ckt** branch circuit identifier,
- Rzer, Xzer** zero sequence branch resistance and reactance,
- Czer** total zero sequence line charging,
- fGzer, fBzer** zero sequence line shunt conductance and susceptance at the from bus end of the branch,

tGzer, tBzer zero sequence line shunt conductance and susceptance at the to bus end of the branch,

Xftype Transformer configuration, entered as an integer based on the following table. Note that the default for a transmission line is 3, since a grounded wye - grounded wye transformer connection has the same equivalent model as a transmission line.

- 0 Wye - Wye
- 1 Grounded Wye - Wye
- 2 Wye - Grounded Wye
- 3 Grounded Wye - Grounded Wye
- 4 Wye - Delta
- 5 Delta - Wye
- 6 Grounded Wye - Delta
- 7 Delta - Grounded Wye
- 8 Delta - Delta

Switched Shunt

SSHUNT busnum numblocks Bzer1 Bzer2 ...

where

busnum terminal bus number,

numblocks number of different zero sequence admittance blocks,

Bzer# zero sequence susceptance for each admittance block, maximum of 8.

Mutual Impedance

MUTIMP from1 to1 ckt1 from2 to2 ckt2 RM XM start1 end1 start2 end2

where

from1, to1 from and to bus numbers of the first mutually coupled branch,

ckt1 circuit identifier of the first mutually coupled branch,

from2, to2 from and to bus numbers of the second mutually coupled branch,

ckt2 circuit identifier of the second mutually coupled branch,

RM, XM zero sequence mutual resistance and reactance,

start1, end1 start and end locations of the section of the first line affected by the mutual coupling, with each point represented as a percentage of the total line length (between 0 and 1),

start2, end2 start and end locations of the section of the second line affected by the mutual coupling, with each point represented as a percentage of the total line length (between 0 and 1).

Exporting Onelines in Different Graphic Formats

Simulator can export oneline diagrams and other graphical displays as bitmaps, metafiles, or jpegs. Select **Export Oneline** from the [Application Button](#). A save file dialog appears where the dropdown menu for files type offers three choices: JPEG, BITMAP, METAFILE. Select the file type and choose the name before clicking the save button.

Exporting an image as a jpeg also requires you to set the compression ratio for the picture. See [Saving Images as Jpegs](#) for more information.

Saving Images as Jpegs

Simulator can save oneline diagrams, bus view displays, and strip charts as jpeg images. To save a oneline diagram or bus view display as a jpeg, select **Export Oneline** from the [Application Button](#). This brings up the save file dialog. Choose jpeg as the file type, type the file name, and press save. This brings up the Jpeg Options Dialog where you decide the picture's resolution. Adjust the resolution control to specify the compression ratio at which to save the diagram as a jpeg. The greater the resolution you specify, the larger the resulting file will be. Click **Save** to save the image or click **Cancel** to terminate the process without saving the image as a jpeg.

To save a strip chart as a jpeg image, right-click on the background of the strip chart and select **Export Image** from the resulting local menu. This brings up the save file dialog. Choose jpeg as the file type, type the file name, and press save. Specify the compression/resolution in the Jpeg Options Dialog. Click **Cancel** to terminate the process, or click **Save** to save the jpeg.

Saving Admittance Matrix and Jacobian Information

The **Save Ybus or Power Flow Jacobian Dialog** is used to store the power system bus admittance matrix (Ybus) and/or the power flow Jacobian in a text format that can be easily read into other programs such as MATLAB®. This dialog is primarily designed for users doing power system analysis research. The dialog has the following fields.

Ybus in MATLAB Format

Filename for Saving Ybus

Enter the name of the file in which to store the Ybus data. The Ybus data is stored using the MATLAB sparse matrix format in the matrix **Ybus**. If the **Include Bus Voltages** field is checked, then the bus voltages are also stored, but in the vector **V**.

Save Ybus in MATLAB Format

Click this button to save the Ybus.

Power Flow Jacobian in MATLAB Format

Jacobian Save File

Enter the name of the file in which to store Jacobian data. The Jacobian is stored using the MATLAB sparse matrix format in the matrix **Jac**.

Jacobian ID Save File

Enter the filename to store the text identifier information. This information is used to translate the bus numbering convention used in the Jacobian and Ybus files with the actual bus number and name in the case.

File Type

Choose the type of MATLAB file you wish to save as. The **MATLAB .M Format** is the more common text format used for directly loading MATLAB information. The **Text for MATLAB Ascii** is for use with MATLAB's ability to read Ascii files. The Ascii file type can be read into MATLAB much faster than the traditional .M files.

Jacobian Form

Select **Rectangular** to store the rectangular form of the Jacobian, or **Polar** to store the polar form of the Jacobian.

Save Jacobian

Click this button to save the Jacobian and object identifier information.

Chapter 4: Building a Oneline Diagram

The oneline objects provide a key means of interacting with the power flow simulation. This chapter provides a reference on how to insert each of the different types of objects shown on the oneline diagram, and an overview of Edit Mode. Note that when you are using the Simulator, you can obtain context sensitive help for oneline objects by positioning the cursor on the desired object and pressing the F1 key.

The following material is included:

- Edit Mode Overview
- Areas, Zones and Owners
- Buses
- Substations
- Generators
- Loads
- Transmission Lines
- Transformers
- Series Capacitors
- Switched Shunts
- Interfaces
- Oneline Links
- Background Objects and Text

Edit Mode Overview

The Edit Mode is used to create and/or modify cases and onelines. You can use the Edit Mode to create a case from scratch or to modify existing power flow cases stored in PowerWorld PWB or AUX files, in the PTI Raw data format, GE PSLF EPC data format, or the IEEE Common Format. New users may wish to view step-by-step tutorials on either

[Creating a New Case from Scratch](#)

[Creating Onelines for an Existing Power Flow Case.](#)

To enter the Edit Mode, select the **Edit Mode** button in the **Mode** ribbon group on any of the ribbon tabs.

A powerful capability of the Simulator is its ability to a case by graphically placing/editing on a oneline diagram. These display objects consist of both power system devices, such as buses, generators, and transmission lines; and that show various system parameters, provide descriptive text, or function as a static background.

Simulator's oneline diagrams illustrate the current state of the components of the power system. Most display objects correspond to records in the underlying power system model, but not all records in the power system model need to have an associated display object. In fact, for large system models, it may be that most of the system will not be illustrated. In such cases, you will want to devote more detail to the more critical areas of the system so as not to clutter the view. Furthermore, it is possible to associate more than one oneline with a single power flow case, and a single oneline may be associated with multiple cases. This great flexibility can prove to be a big time-saver. Please see [Relationship Between Display Objects and the Power System Model](#) for a more thorough discussion.

Edit Mode General Procedures

In order to simplify the process of graphically constructing a power flow case, Simulator's drawing interface obeys the following conventions for most objects:

Inserting a New Object

- Go to the [Draw](#) ribbon tab and use the menus on the [Individual Insert](#) ribbon group to add a single oneline object type you would like to add to the oneline.
- Left-click on the location on the display where you would like to position the object.
- Once the object is placed, Simulator displays a dialog box that allows you to specify various options for the object.
- If desired, use all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the appearance of the object once it has been placed.

Moving, Resizing, and Rotating an Existing Object

- Select the object by clicking on the object with the left mouse button. Handles are displayed around the object to indicate it has been selected.
- To move the position of the object, place the mouse anywhere on the object except at a handle location. Then drag the object around the screen by holding the left mouse button down.
- To change the size of an object using the mouse, first select the object. Then place the mouse on one of the object's resizing (white) handles. The cursor will change to either a horizontal, vertical or diagonal two-headed arrow shape. Then drag the mouse to change the object's size. You can also specify the size of most objects using their dialog boxes.
- To rotate an object, first select the object. Then place the mouse over the rotation (green) handle in the upper left corner of the selected object. The cursor will change to a rotation symbol. Clicking and dragging will then rotate the object. Note that if an object (e.g., a pie chart or transmission line) does not have a rotation handle in the upper left, the object cannot be rotated.

Viewing/Modifying Object Parameters

- To view and/or change the options associated with a single object, right-click on the object. This either displays the object's dialog box directly, or it displays the object, from which you can elect to see the object's dialog box.

Selecting Several Objects to Modify Their Appearance

Hold down the Shift key while clicking objects on the screen to select several objects at once. You may then change the objects' attributes, such as , by using all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

To move the objects that are selected, click and hold down the left mouse button on any of the selected objects, drag the selected objects to a new location, and release the mouse button to place them. You can also move the selected objects by holding the SHIFT key and simultaneously using the UP, DOWN, LEFT, or RIGHT arrow keys (or use the Home, End, Page Up and Page Down keys to move objects).

To select a set of objects use the many options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects using a [rectangle, ellipse or polygon selector](#). You can also select only objects that meet some given criteria by using [Select by Criteria](#). As an example, you can

use Select By Criteria to select all the 345 kV transmission lines in a case. See [Select](#) ribbon group for more detailed help.

Changing An Object's Screen Appearance

Use all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the screen appearance of either a selection of objects or the entire display.

Draw Ribbon Tab

The Draw ribbon tab is primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in [Edit Mode](#). The Draw ribbon tab is shown below.



The Draw ribbon consists of seven ribbon groups described as follows

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. For detailed help see the [Quick Insert](#) ribbon group. If you would instead like help on inserting a single object on your oneline diagram see the see the [Individual Insert](#) ribbon group topic.

Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert](#) ribbon group. However, for complete access to all objects including those that can not be "quick - inserted", see the help under the [Individual Insert](#) ribbon group topic.

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. For more detailed help see the [Selection](#) ribbon group topic for more detailed help.

Formatting Ribbon Group

The Format ribbon group contains options for formatting the selected objects on your active oneline. For more detailed help see the [Formatting](#) ribbon group topic for more detailed help.

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline display objects. For more detailed help see the [Clipboard](#) ribbon group topic for more detailed help.

Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

Oneline Display Options

The [Oneline Display Options Dialog](#) allows you to customize the appearance of the presently selected oneline diagram. To view this dialog, either select **Oneline Display Options** from the **Oneline Options** ribbon group on the [Options](#) ribbon tab, or choose **Oneline Display Options** from the oneline's [local menu](#) . Please see the [Oneline Display Options Dialog](#) help for more information.

Anchored Objects

While in [Edit Mode](#), Simulator allows certain objects to be attached, or *anchored*, to another object, called the *anchor*. When an object that functions as an anchor is moved, all objects that are anchored to it will move with it. This feature can be very useful when you move objects around the oneline diagram in Edit Mode.

Anchoring has the property of "stacking" in Simulator. In other words, one object is anchored to another, which is in turn anchored to yet another. The best way to describe this is by example. A generator text field can be anchored to a generator. The generator, in turn, can be anchored to its terminal bus. If you move the terminal bus, both the generator and its anchored fields also move with the bus. However, if you just move the generator itself, only the generator fields will move with it. The bus and all other objects anchored directly to the bus remain in their original location.

If anchors are deleted it is generally necessary to reset anchors on anchored objects go to the [Draw](#) ribbon tab and choose **Refresh Anchors** on the [Formatting](#) ribbon group.

There are several types of anchored objects:

[Buses](#)

[Loads](#), [generators](#), [switched shunts](#), [bus fields](#), [interfaces](#), [transformers](#), [transmission lines](#) and [voltage gauges](#) may be anchored to their associated bus. When the anchor bus is moved, these anchored objects will move with it.

[Substations](#)

Similar to buses, any [loads](#), [generators](#), [switched shunts](#), [bus fields](#), [interfaces](#), [transformers](#), and [transmission lines](#) may be anchored to their associated substations. When the anchor substation is moved, these anchored objects will move with it.

[Generators](#)

[Generator fields](#) may be anchored to their associated generator. When the anchor generator is moved, these anchored fields will move with it.

[Loads](#)

[Load fields](#) may be anchored to their associated load. When the anchor load is moved, these anchored fields will move with it.

[Switched Shunts](#)

[Switched shunt fields](#) may be anchored to their associated switched shunt. When the anchor switched shunt is moved, these anchored fields will move with it.

[Area/Zone/Super Area Objects](#)

[Interfaces](#) can be anchored to area/zone/super area objects.

[Lines and Transformers](#)

[Circuit breakers](#), [line flow pie charts](#), and [line fields](#) may be anchored to their associated line/transformer. When the line/transformer is moved, these anchored objects will move with it.

[Interfaces](#)

[Interface fields](#) and [interface pie charts](#) can be anchored to their associated interface.

Inserting and Placing Multiple Display Fields

Simulator provides a convenient method of adding display fields to a variety of oneline display objects and placing them in default positions relative to the display object. Unlike the field placement implemented by selecting **Field** from the **Individual Insert** ribbon group on the **Draw** ribbon tab, the method described here allows you to place multiple fields around a display object in a single operation. This option is available for several different types of objects, including buses, generators, lines and transformers, shunts, and loads. The option can always be accessed by right clicking the display object of interest and selecting the **Add New Fields Around ...** option from the resulting local menu. This brings up the **Insert New Fields around Selected Objects Dialog**. This dialog is divided into several tabs by object type. Only tabs that correspond to the type of objects you have selected will be available.

The tabs for inserting and placing multiple display fields for the various display objects (buses, generators, lines/transformers, shunts, and loads) are virtually identical in content. The tab illustrates the possible locations of the display fields for the various orientations of the display object. For example, since buses may be oriented either horizontally or vertically, the dialog shows how each of the eight possible bus fields would be positioned for each of the two orientations. Generators, loads, and shunts each have four possible orientations, so the dialog identifies the locations for the possible fields for each of the four orientations. Transmission lines and transformers can assume only one orientation, and the dialog will thus show the possible field locations for that single orientation.

Each field location is identified on the illustrations with a label of the form Pos #. In order to modify the settings, move your mouse over the position on the dialog you want to change and click. This will bring up the appropriate Field Options Dialog such as the [Bus Field Options](#), [Generator Field Options](#), [Load Field Options](#), [Switched Shunt Field Options](#), or the [Line Field Options](#). Simply select the field you want and choose **OK**. If you would like to set a default field to "none", click **Remove Field** instead of **OK**.

Click **OK** to implement your choices for field additions and placement, or click **Cancel** to discard the changes.

This help topic has gone over how to add new fields to existing display objects. It should be noted from this discussion that all objects have an associated set of default fields that will be added to the oneline when the objects are originally inserted. You may redefine the default fields selecting **Default Drawing Values** from the **Quick Insert** ribbon group on the **Draw** ribbon tab and modifying the [Default Drawing Options Dialog](#).

Area Display Objects

Area records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by area.

To insert an Area object, select **Aggregation > Area** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Area object should be placed. This will open the [Area Display Options](#) dialog, which will allow you to choose the area information to display, and set parameters for the displayed object. It is important to note that area records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new area record to the load flow case, you need to change the area designation of a device in the load flow case to a new area number, or you need to insert a new area from the [Area Records](#).

Area objects can also be added by using the [Area insert palette](#).

Area Display Options Dialog

When you insert an area display object on a oneline diagram, Simulator opens the Area Display Options dialog. This dialog is used to control various display and identity attributes of the area display object. The dialog contains the following fields:

Number

This dropdown box lists the number of all areas in the case. Use this control to associate the display object with the correct area.

Name

If you would prefer to search through areas by name rather than by number, use the *Name* dropdown box to see a list of names of all the areas in the case.

Show Record Type Prefix

Check this box if you wish to place the prefix **Area** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Area Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle, rounded rectangle, or an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Zone Display Objects

Zone records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by zone.

To insert a Zone object, select **Aggregation > Zone** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Zone object should be placed. This will open the [Zone Display Options](#) dialog, which will allow you to choose the zone information to display, and set parameters for the displayed object. It is important to note that zone records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new zone record to the load flow case, you need to change the zone designation of a device in the load flow case to a new zone number, or you need to insert a new zone from the [Zone Records](#).

Zone Display Options Dialog

When you insert a zone display object on a oneline diagram, Simulator opens the Zone Display Options dialog. This dialog is used to control various display and identity attributes of the zone display object. The dialog contains the following fields:

Number

This drop down box lists the number of all zones in the case. Use this control to associate the display object with the correct zone.

Name

If you would prefer to search through zones by name rather than by number, use the *Name* drop down box to see a list of names of all the zones in the case.

Show Record Type Prefix

Check this box if you wish to place the prefix **Zone** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Zone Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle, rounded rectangle, or as an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Super Area Display Objects

Super area records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by super areas.

To insert a Super Area object, select **Aggregation > Super Area** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Super Area object should be placed. This will open the [Super Area Display Options](#) dialog, which will allow you to choose the super area information to display, and set parameters for the displayed object. It is important to note that super area records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new super area record to the load flow case, you need to insert a new super area from the [Super Area Records](#).

Super Area Display Options Dialog

When you insert a super area display object on a oneline diagram, Simulator opens the Super Area Display Options dialog. This dialog is used to control various display and identity attributes of the super area display object. The dialog contains the following fields:

Number

This field is unused for super area objects. Super areas are identified by a unique super area name instead.

Name

Use the *Name* dropdown box to see a list of names of all the super areas in the case.

Show Record Type Prefix

Check this box if you wish to place the prefix **Super Area** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Super Area Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle, a rounded rectangle, or an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Owner Display Objects

Owner records in the load flow data can be displayed as objects on a Simulator online diagram. This can be useful for building a diagram on which you also want to include summary objects by owner, in which information about the generation and load of the owner are indicated with the object.

To insert an Owner object, select **Aggregation > Owner** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a online diagram in the location where the owner object should be placed. This will open the [Owner Display Options](#) dialog, which will allow you to choose the owner information to display, and set parameters for the displayed object.

Owner Display Options Dialog

When you insert an owner display object on a oneline diagram, Simulator opens the Owner Display Options dialog. This dialog is used to control various display and identity attributes of the owner display object. The dialog contains the following fields:

Number

The owner number.

Name

The owner name.

Show Record Type Prefix

Check this box if you wish to place the prefix **Owner** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the **Owner** prefix.

The remainder of the choices presented on the *Owner Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle or as an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Area Fields on Onelines

Area fields are used to show various values associated with a particular area of the power system. Right clicking on the area field displays the [Area Field Dialog](#).

Edit Mode

To enter a new area field, first select **Field > Area Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click on or near the area object or a bus in the area for which you want to add a field. This calls up the [Area Field Dialog](#). Verify that the area number is correct. By default, this value is the number of the area associated with the closest area object, or if there are no area objects, the closest bus. Enter the total number of digits the field should display as well as the number of digits to the right of the decimal point. Depending on what the field is designed to display, you may need to enter an additional area number. Finally, select the field type. Click **OK** to save the field or **Cancel** to abort the operation.

With most types of area fields, an **Area Number** of 0 is valid and defines the field as showing values for the entire system.

To modify the parameters of an existing area field, position the cursor anywhere on the area field and right-click. This again brings up the [Area Field Dialog](#). Use the [Formatting](#) options to change various display attributes for the area field, including its font and background color.

Zone Fields on Onelines

Zone field objects are used to show different values associated with zones and the system. This dialog is used to view (and in a few cases modify) the parameters associated with these zone fields. To reach this dialog, select **Field > Zone Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and click the desired field location on a oneline diagram in Edit Mode, or right-click on an existing zone field on a oneline diagram.

Zone Number

Zone number associated with the field. When you insert fields graphically, this field is automatically set to the zone number associated with the closest bus on the oneline. With most types of zone fields, a *Zone Number* of 0 is valid and defines the field as showing values for the entire system.

Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Fields

Total number of digits to show in numerical fields.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point in numerical fields.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Other Zone Number

Some of the fields, such as **MW Flow to Other Zone**, require that a second area be specified. If applicable, enter the second (other) zone here.

Delta per Mouse Click

This value is used only with the **Sched Flow to Other Zone** field type. When there is a nonzero entry in this field, and the field type is **Sched Flow to Other Zone**, a spin button is shown to the right of the zone field. When the up spin button is clicked, the flow to the other zone is increased by this number of MW; when the down button is clicked, the scheduled flow is decreased by this amount.

Field Value

Shows the current output for the zone field. Whenever you change the **Type of Field** selection, this field is updated.

For the **Sched Flow to Other Zone** field type only, you can specify a new value in MW. Exports are assumed to be positive.

Field Prefix

An optional string that precedes the field value.

Anchored

If the field is associated with a Zone Object on the diagram, the field can be anchored to the object so that if the object gets moved on the diagram, the field will move with it.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of zone field to show.

Name

Zone name.

Number

Zone number (1 - 999).

MW Load, Mvar Load

If the zone number is nonzero, then these fields show Total MW or Mvar load for the zone. If the zone number is zero, these fields show the total load in the entire system.

MW Generation, Mvar Generation

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar generation for the zone. If the *Zone Number* is zero, these fields show the total generation in the entire system.

MW Shunts, MVR Shunts

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar shunt compensation for the zone. If the *Zone Number* is zero, these fields show the total shunt compensation in the entire system.

MW Flow to Other Zone, Mvar Flow to Other Zone

Total MW or Mvar flow from the zone specified in the *Zone Number* field to the zone specified in the *Other Zone Number* field. The *Zone Number* field must correspond to a valid zone. If the *Other Zone Number* field is zero, this field shows the zone's total MW or Mvar exports.

MW Losses, MVAR Losses

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar losses for the zone. If the *Zone Number* is zero, these fields show the total real or reactive losses in the entire system.

Load Schedule Multiplier

Indicates the current value of the MW multiplier applied to the zone's loads.

Select a Field

Allows you to choose any other field associated with a zone not explicitly listed as an option in the Type of Field selection.

Select *OK* to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

Super Area Fields on Onelines

To display certain information about a super area, such as MW Load or MVAR losses, insert a super area field. This can be done in Edit Mode by selecting **Field > Super Area Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will bring up the [Super Area Field Information](#). From here you can choose which super area to describe, how many digits in the field, and how many digits to the right of the decimal. There are also 12 different field options to choose from. If a field value is not defined, question marks will be shown.

Owner Fields on Onelines

Owner field objects are used to show different values associated with owners. This dialog is used to view and modify the parameters associated with these fields. To insert an Owner field, click on **Field > Owner Field** in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Owner Number

Owner number associated with the field. When you insert fields graphically, this field is automatically set to the owner number associated with the closest bus on the oneline.

Find by Number

To switch to a different owner in the field options dialog, you can enter the number in the **Owner Number** field, and press the **Find by Number** button to update the dialog with information for the new owner.

Owner Name

Name of the owner whose information is presently being displayed in the dialog.

Find by Name

To switch to a different owner in the field options dialog, you can enter the name in the **Owner Name** field, and press the **Find by Name** button to update the dialog with information for the new owner.

Total Digits in Fields

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Prefix

An optional string that precedes the field value.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Field Value

Shows the current output for the owner field. Whenever you change the **Type of Field** selection, this field is updated.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of owner field to show. You can choose from one of the six default fields, or click on the **Find Field...** button to choose from a list of all available owner fields.

Bus Display Objects

In power system analysis, the term "bus" is used to refer to the point where a number of electrical devices, such as lines, loads or generators, join together. On the oneline diagram, buses are usually represented with either a thick horizontal line or a thick vertical line. The bus thickness and color can be customized.

Right-clicking on the bus will display its local menu. The local menu offers you the chance to view the corresponding [Bus Dialog](#), the [Quick Power Flow List](#), and the [Bus View Display](#). When the application is in Edit Mode, the local menu will also allow you to add bus fields to the bus and to insert any undrawn buses connected to the selected bus. [Bus Fields](#) are often placed close to the bus to indicate its voltage magnitude, voltage angle, and other relevant information.

Edit Mode

To add a new bus to the case graphically, follow this simple procedure:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group of the [Draw](#) ribbon tab. This prepares Simulator to insert a new bus.
- Left-click on the oneline background at the location where you want to place the new bus. This invokes the [Bus Option Dialog](#).
- Use the Bus Option Dialog to specify the number, name, size, thickness, orientation, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to it. Every bus must have a unique number.
- The Bus Option Dialog will include a message just above the Bus Number field indicating that a new bus will be inserted into the power system data model. This will appear as long as the Bus Number that is selected does not already exist in the power system case.
- Click **OK** on the Bus Option Dialog to finish creating the bus and to close the dialog. If you do not wish to add the bus to the case, click **Cancel**.

If you are simply adding a symbol to the oneline diagram for a bus that has already been defined in the case, many of the parameters you are asked to specify in step three will be filled in for you.

To modify the parameters for an existing bus, position the cursor on the bus and right-click to invoke the bus' local menu. From the local menu, choose **Bus Information Dialog** to view the associated [Bus Dialog](#). You may change any of the parameters specified there. When a bus' number is changed, the bus numbers associated with all of the devices attached to that bus are also automatically changed. To renumber several buses simultaneously, please see [Bus Renumbering Dialog](#).

To modify any aspect of a bus' appearance, first select the bus, and then click any of the format buttons found in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. You can change the length of the bus (but not its thickness) by dragging the bus' resizing handles.

In order to delete an existing bus after selecting it, use either the **Cut** button, found in the [Clipboard](#) ribbon group on the [Draw](#) ribbon tab, to preserve a copy of the bus on the Windows clipboard, or **Delete** button, found in the [Clipboard](#) ribbon group on the [Draw](#) ribbon tab, to remove the bus without copying it to the clipboard. You will be asked whether you want to remove both the display object and its associated bus record, or merely the display object, leaving the bus in the power flow model. If you will never be deleting a record from the power system model, you may also choose the option labeled **Always Delete Objects Only**. Be careful when deleting existing buses with attached devices. An error will occur during validation if you do not also delete the attached devices or attach them to other buses.

Bus Fields on Onelines

Bus field objects are used primarily to indicate various quantities associated with bus devices. Furthermore, some bus field types, which are distinguished by an integrated spin button, may be used to easily change bus device properties.

Run Mode

Right clicking on a bus field gives you the option to open the [Bus Field Dialog](#) or the [Bus Information Dialog](#).

Edit Mode

Simulator offers two options for adding bus fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Bus Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then select the bus to which you want to add the field. This invokes the [Bus Field Dialog](#). Enter the bus number associated with the device (the default is the closest bus to the field), the total number of digits to show, and the number of digits to the right of the decimal point. An optional Field Prefix can be used. Choosing to Include Suffix will include relevant units for the type of field selected. The field can be Anchored to the bus that it represents and the field can be rotated by a specified Rotation Angle in Degree. Finally, select the type of field to show.

The second approach for adding new bus fields entails right-clicking the bus and selecting **Add New Fields Around Bus** from the resulting local menu. See [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing bus field, position the cursor anywhere on the object and right-click. This brings up the [Bus Field Dialog](#). Choose any of the format buttons on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change various display attributes of the field, including its font and background color.

Old Voltage Gauges

Voltage gauges provide a way to visualize the voltage of a bus relative to its high- and low-voltage alarm limits. A gauge looks very much like a thermometer. As the temperature changes, the height of the mercury in the thermometer moves up and down. One reads the temperature measured by the thermometer by noting the marking that matches the top of the mercury. Voltage gauges in Simulator work the same way. A voltage gauge has three markings on its side, each of which identifies a key per-unit voltage level. Specifically, these three markings locate the minimum, maximum, and target per-unit voltages. Often, the target voltage level is the nominal voltage, but this is not a requirement. Inside the gauge is a filled region. The default color of the filled region is blue, but this can be changed. When you create the voltage gauge, you associate it with a bus, and you specify its fill color and its minimum, maximum, and target voltage levels. Once the gauge has been placed on a display, it will reveal changes in its associated bus's voltage by varying the height of its filled region. This tool was introduced to provide an alternative to voltage contours to show the variation of voltage across a region. [Contouring](#) can reveal the variation of only a single quantity at a time. For example, it is impossible to contour bus voltage magnitude and bus voltage phase angle simultaneously. Voltage gauges are helpful because they allow you to show the voltage profile superimposed on a contour of some other quantity.

To add a voltage gauge to a display, switch to Edit Mode and select **Pies / Gauges > Old Gauges > Bus** from the **Individual Insert Ribbon Group** on the [Draw](#) ribbon tab. The cursor will become a crosshair. Click on the oneline diagram where you would like the new voltage gauge to appear. The [Voltage Gauge Options Dialog](#) will appear. Use this dialog to define the minimum, maximum and target voltages for the voltage gauge, as well as its fill color and whether it should be anchored to its associated bus. After you click the "OK" button, the Voltage Gauge Options Dialog will close, and the new voltage gauge will appear.

Once a gauge has been placed on the oneline, the height of its filled region will change as the voltage of its associated bus changes. To modify any of the characteristics of the gauge, such as its key voltage levels, fill color, and anchor setting, simply right-click on the bus voltage gauge to open the Voltage Gauge Options Dialog again.

Old Voltage Gauge Options Dialog

The Voltage Gauge Options Dialog is used to define and configure a [bus voltage gauge](#). A bus voltage gauge is associated with a particular bus and reveals the bus's voltage relative to specified minimum, maximum, and target per-unit voltage levels. The height of the colored column in the voltage gauge indicates the bus's voltage relative to these markings.

The dialog has the following controls:

Number, Name, and Find

Use the **Number** and **Name** dropdown boxes to identify the bus to which you want the gauge to correspond. Select a bus number from the Number dropdown box to identify the bus by number, and a bus name from the Name dropdown box to identify a bus by name. It may be more convenient to press the **Find** to open the [Find Dialog](#), which allows you to specify a bus by either name or number using wildcards. When you first open the Bus Voltage Gauge Dialog, the bus name and number will correspond to the bus object that was closest to the point where you clicked.

Minimum, Target, and Maximum

Use these three spin edit boxes in the **Voltage levels** group box to specify the minimum, maximum, and target per-unit voltage levels. These settings determine where on the gauge its three markings will be drawn.

The Minimum and Maximum value will be taken from the Limit Monitoring settings for the current bus, unless you uncheck the option "*Set Limits According to Current Limit Monitoring Settings*".

Fill color

The Fill color box reveals the color that will be used to paint the filled region of the gauge. Click on the Fill color box to open a Color Dialog, which you may then use to specify a different color.

Anchored

A bus voltage gauge is said to be [anchored](#) if, when you move its associated bus object, it moves with it. Check the Anchored check box to ensure that the gauge will move with its associated bus. Otherwise, when you move its associated bus object, the gauge will stay in its current position.

OK, Help, and Cancel

Click OK to finalize your settings. This will create a new voltage gauge object if you are trying to create one from scratch, or it will modify the appearance and settings of an existing one if you have chosen to modify one that has already been defined. Click Cancel to dispose of your changes. Click the Help button to reveal this help text.

Substation Display Objects

Substations in Simulator define a group of buses that are closely connected. Each bus can belong to either one substation or no substation (called unassigned). By default in Simulator, all buses are not assigned to a substation. This is done because all traditional text file power flow formats do not include information regarding a bus' substation.

Substations are represented on the oneline as a rectangle with the name of the substation inside it. Other information about the substation is also displayed on the rectangle:

- The upper left corner displays a generator symbol if generation exists in the substation.
- The upper right corner displays a load symbol if load exists in the substation.
- The lower right corner displays a shunt symbol if shunts exist in the substation.
- The lower left corner displays the number of buses inside the substation.
- The lower middle displays the maximum voltage level in the substation.

You can also customize the size, colors, and font name and style of a substation object. Note however that the font size of the substation object is automatically changed by Simulator as you change the size of the rectangle.

Right-clicking on the substation will display its local menu. The local menu offers you the chance to view the corresponding [Substation Information Dialog](#) and the [Substation View Display](#).

Substation Fields on Onelines

To display certain information about a substation, such as MW Load or MVAR losses, insert a substation field. This can be done in Edit Mode by selecting **Fields > Substation Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will bring up the [Substation Field Options](#) dialog. From here you can choose which substation to describe, how many digits in the field, and how many digits to the right of the decimal. There are also 8 different field options to choose from. If a field value is not defined, question marks will be shown.

Generator Display Objects

Generators are represented on the oneline as circles with a "dog bone" rotor inside. Multiple generators at a bus are allowed, with each being distinguished by a unique character identifier.

Each generator symbol (except that corresponding to the slack) is equipped with a circuit breaker that can be used to change the status of the generator. You may toggle the generator status by clicking on the circuit breaker while in Run Mode.

[Generator fields](#) are often placed close to the generator on the oneline to indicate the generator's MW/Mvar output.

Run Mode

When [animation](#) is active, the default flow of the arrows emerging from the generator is proportional to its MW output. You can customize the appearance of this flow using the Animated Flows Tab of the [Oneline Display Options Dialog](#).

Right clicking on the generator brings up the generator submenu. This menu is used to display a variety of information about the generator, including

- [Generator Information Dialog](#)
- [Input-output curve](#)
- [Fuel cost curve](#)
- [Incremental cost curve](#)
- [Heat-rate curve](#)
- [Reactive capability curve](#)

Edit Mode

To add a new generator to the case, select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, place the cursor on the bus where you would like to attach the generator, and click with the left mouse button. This calls up the [Generator Dialog](#). The bus number is automatically determined from the bus to which you attached the generator. The ID field contains a alphanumeric ID used to distinguish multiple generators at a bus. The default is '1'.

Enter the size, the thickness of the lines (in pixels) used to display the device, orientation, and other parameters for the generator. Each generator automatically contains a switch for connecting or disconnecting the device in Simulator. Select OK to add the generator. If you do not want to add the generator to the case, select Cancel.

To modify the parameters for an existing generator, position the cursor on the generator and right-click. This again brings up the Generator Dialog. You can then change any parameter (be careful in renumbering an existing generator). Use the [Formatting](#) ribbon group commands to change the color, line thickness, and other display parameters.

Generator Fields on Onelines

Generator field objects are used primarily to indicate various quantities associated with generation devices. Furthermore, some generator field types, which are distinguished by an integrated spin button, may be used to change generation device properties.

Run Mode

For generator fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a generator field gives you the option to open the [Generator Field Dialog](#) or the [Generator Information Dialog](#).

Edit Mode

Simulator offers two options for adding generator fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Generator Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and click near generator for which you want to add the field. This invokes the [Generator Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest generator to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show. For generator actual MW and Setpoint MW types and load MW and Mvar fields, specify a nonzero value in the *Delta per Mouse Click* to design a bus field with an integrated spin control. The Gen AGC Status field is used to display the automatic generation control status of the generator. The user can toggle this status in Simulator by clicking on the field. Likewise, the Gen AVR Status field is used to display the automatic voltage regulation status of the generator. Again, the user can toggle this status by clicking on the field.

The second approach for adding new generator fields entails right-clicking the bus and selecting *Add New Fields Around Generator* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing generator field, position the cursor anywhere on the object and right-click. This brings up the [Generator Field Dialog](#). Choose , or  from the [Formatting](#) ribbon group to change various display attributes of the field, including its font and background color.

Load Display Objects

Simulator models aggregate load at each system bus. Multiple loads at a bus are allowed. Each load object on the oneline comes equipped with a circuit breaker. The status of the load corresponds to the status of its circuit breaker. A circuit breaker is closed if it appears as a filled red square, and it is open if it appears as a green square outline. In Run Mode, you may toggle the status of the load by clicking on its associated circuit breaker.

[Load fields](#) are often placed close to the loads on the oneline to indicate their MW/Mvar value.

Run Mode

When [animation](#) is active, the flow of the arrows into the load is proportional to its current MW load. You can customize the appearance of this flow using the Animated Flows Tab of the [Oneline Display Options Dialog](#).

Right clicking on a load (bus) field gives you the option to open the [Load Field Dialog](#) or the [Load Dialog](#).

Edit Mode

To add a new load to the case, select **Network > Load** from [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, select the bus to which you want to attach the load with the left mouse button. This calls up the [Load Dialog](#). The bus number is automatically determined from the bus to which you attached the load. The ID field contains a two-character ID used to distinguish multiple loads at a bus. The default ID is 1.

Enter the size, the thickness of the lines [in pixels] used to display the device, the orientation, and the base MW and Mvar load values for the device. Usually, only the Constant Power fields are specified as nonzero. The Constant Current and Constant Impedance fields are used to specify loads that vary with voltage. Constant current loads vary proportionally with bus voltage, while constant impedance loads vary with the square of the voltage. Specify the constant current and constant impedance values assuming one per-unit voltage.

Select OK to add the load. If you do not want to add the load to the case, select Cancel.

To modify the parameters for an existing load, position the cursor on the load and right-click. Select *Load Information Dialog* from the local menu to invoke the [Load Dialog](#). You can then change any parameter as desired. You can select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the drawing parameters of the load.

Load Fields on Onelines

Load field objects are used primarily to indicate various quantities associated with load devices. Furthermore, some load field types, which are distinguished by an integrated spin button, may be used to change load device properties.

Run Mode

For load fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a load field gives you the option to open the [Load Field Dialog](#) or the [Load Information Dialog](#).

Edit Mode

Simulator offers two options for adding load fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Load Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then select the load to which you want to add the field. This invokes the [Load Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest load to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show.

The second approach for adding new generator fields entails right-clicking the bus and selecting *Add New Fields Around Load* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing load field, position the cursor anywhere on the object and right-click. This brings up the [Load Field Dialog](#). Choose  or  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change various display attributes of the field, including its font and background color.

Transmission Line Display Objects

Transmission lines are represented on the onelines using multiple segment lines drawn between buses. Transmission lines may be equipped with circuit breakers that can be used to change the line's status. You can also add [pie charts](#) and [line fields](#) to transmission lines to indicate how heavily loaded the line is. The appearance of transmission lines, including line thickness and color, may also be customized.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a transmission line displays the line's local menu, from which you can choose to inspect the [Line/Transformer Dialog](#).

Edit Mode

To add a new transmission line to the case, first select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transmission line (the "from" bus) and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete adding a new line, place the cursor on the second bus for the line (the "to" bus) and double-click with the left mouse button. This calls up the [Line/Transformer Dialog](#). The "from" and "to" bus numbers are set automatically provided the line starts and ends on existing buses. If there is just one line between the buses, the circuit number should be "1." For multiple lines between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the transmission line. Enter the per unit (100 MVA base) resistance, reactance, total charging susceptance (that is B not B/2) for the line, and an MVA rating. Select OK to add the line. If you do not want to add the line to the case, select Cancel.

To modify the parameters for an existing line, position the cursor anywhere on the line and right-click. This brings up the [Line/Transformer Dialog](#), which you can use to change various line parameters.

To change the physical appearance of the line (i.e. line color, line thickness, etc.), make use of the tools in the [Formatting](#) ribbon group.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex clicking and holding the left mouse button down on the vertex, dragging it to a new location, and then releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

Line Fields on Onelines

Line field objects are used to show values associated with lines, transformers and dc transmission lines.

Run Mode

Right clicking on the line field gives you the option to open the [Line Field Dialog](#) or the [Line Information Dialog](#).

Edit Mode

To enter a new line field, select **Field > Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and select the line, transformer, or dc line to which you want to add the field. The [Line Field Dialog](#) will open. Enter the *near* and *far* bus number associated with the device (the default values for these fields correspond to the device on which you clicked) and the circuit number of the device. The field will display the flow value at the *near* end of the device. Enter the total number of digits to show on the screen and the number of digits to the right of the decimal point. Finally, select the desired field type.

To modify the parameters of an existing line field, position the cursor anywhere on the object and right-click. This again brings up the [Line Field Dialog](#). Select  from the [Formatting](#) ribbon group on the Draw ribbon tab to change many of the line field's display attributes.

Another way to add line fields to the oneline entails right-clicking the line and selecting *Add New Fields Around Line* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

Circuit Breakers on Onelines

Circuit breakers are used to open or close transmission lines and transformers. They are also used to place generators, loads and switched shunts in or out of service. By default, closed circuit breakers are shown as solid red squares, while open circuit breakers are shown as a green square outline. The color and filled or unfilled properties of circuit breakers can be modified in the [Online Display Options](#) under [Display Object Options](#). Only those circuit breakers that are set to use the default shape will abide by the shape specified with the Online Display Options. Circuit breakers for generators, loads, and switched shunts always use the rectangle shape. Circuit breakers that are automatically inserted for transmission lines and transformers will use the default shape unless the user specifies a different shape.

To change the status of a breaker, left-click on the breaker while in run mode. A circuit breaker directly controls the status of its associated display object. One breaker is shown on the line connecting a generator, load, or shunt to its associated bus. Two breakers are shown by default on transmission lines and transformers. Opening either of the line's circuit breakers opens the transmission line or transformer; you do not have to open circuit breakers at both ends of the line.

Note: Circuit breakers cannot be placed on dc transmission lines.

Edit Mode

By default, when you add a new display object to a oneline diagram, the necessary circuit breakers are automatically added at each end of the branch.

To add a new circuit breaker to the oneline, select **Indication > Circuit Breaker** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then click the line to which you want to add the breaker. The Circuit Breaker Options dialog will appear with the *from* and *to bus* numbers, and the circuit number automatically set to identify the line you selected. Specify the size for the switch and its initial status, as well as whether it will be anchored to the line so that it will move with the line. Click **OK** to add the circuit breaker, or click **Cancel** to abort the process. To add more circuit breakers to the line, simply repeat this procedure.

The shape of a branch circuit breaker can be user-specified on the Circuit Breaker Options dialog. If the option to *Use Default* is selected for the **Shape**, the shape is dictated by the default circuit breaker shape specified with the [Online Display Options](#) under the [Display Object Options](#). If a non-default shape is specified that requires an orientation, the **Orientation** option will be enabled.

To modify the parameters for an existing breaker, position the cursor anywhere on the device and right-click. This invokes the Circuit Breaker Options Dialog, from which you can change many of the breaker's parameters. Additional display settings for the circuit breaker can be accessed by selecting  from the [Formatting](#) ribbon group.

You can toggle the status of Transmission Line and Transformer circuit breakers while in Edit Mode by right-clicking on the associated line or transformer and selecting **Open Line** or **Close Line** from the local menu.

Run Mode

Circuit Breaker status can be toggled in Run Mode by Left-Clicking on a breaker to place its associated object either in or out of service. If the simulation is currently running, any effects of changing a circuit breaker's status are immediately shown on the oneline.

Line Flow Pie Charts on Onelines

The line flow pie charts are used to indicate the percentage MVA, MW, or Mvar loading of a transmission line or a transformer. The degree to which the pie chart is filled shows how close the device is to its limit (provided the device has a nonzero limit). A line flow pie chart becomes completely filled when the device's flow meets or exceeds 100% of its rating.

For pie charts for objects that are not associated with line flows, see [Pie Charts / Gauges](#).

Use the *Pie Chart Options* tab of the [Online Display Options Dialog](#) to customize various attributes of all pie charts. The tab allows you to define a warning level at which the size and color of the pie charts will change to a size and color you specify. The tab also allows you define a limit percent as well as the size and color to which to change the pie charts when their corresponding devices violate their limits. You can also specify whether the pie charts should reveal total power flow (MVA), real power flow (MW), or reactive power flow (MVR). The oneline display options dialog can be invoked either by selecting **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab or by right-clicking on the background of the oneline diagram and selecting *Online Display Options* from the resulting local menu.

Right clicking on the pie chart displays the Line Flow Pie Chart Options Dialog. This dialog allows you to view the *from* and *to bus* numbers and the circuit number of the line/transformer associated with the pie chart. You can change the pie chart's size and the MVA rating of the line/transformer associated with the pie chart.

Edit Mode

To enter a new line flow pie chart, select **Pies/Gauges > Line Flow Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab [Ref679059392](#) and click on the line or transformer to which you want to add the pie chart. This opens the Line Flow Pie Chart Options Dialog box. Enter the near and far bus number associated with the device (these fields default to the terminal bus numbers of the device on which you clicked), the circuit number of the device, and the desired size of the pie chart. The field will display the flow value at the *near end* of the device. Enter the size of the device. Select **OK** to insert the line flow pie chart. Otherwise, select **Cancel**.

To modify the parameters of an existing line flow pie chart, position the cursor anywhere on the object and right-click. This again brings up the Line Flow Pie Chart Options Dialog box.

Line Flow Gauges

Line flow gauges provide a way to visualize the flow of a transmission line or transformer relative to its thermal rating. A gauge looks very much like a thermometer. As the temperature changes, the height of the mercury in the thermometer moves up and down. One reads the temperature measured by the thermometer by noting the marking that matches the top of the mercury. Line flow gauges in Simulator work the same way. A line flow gauge has two markings on its side, one for the designated minimum flow, and one for the designated maximum flow (the branch's rating). Inside the gauge is a filled region. The default color of the filled region is blue, but this can be changed. When you create the line flow gauge, you associate it with a transmission element, and you specify its fill color and its minimum and maximum flow levels. Once the gauge has been placed on a display, it will reveal changes in its associated branch's flow by varying the height of its filled region. This tool was introduced to provide an alternative to [line flow pie charts](#).

To add a line flow gauge to a display, switch to Edit Mode and select **Pies / Gauges > Old Gauges > Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the oneline diagram where you would like the new line flow gauge to appear. The [Line Flow Gauge Options Dialog](#) will appear. Use this dialog to define the minimum and maximum flows for the line flow gauge, as well as its fill color and whether it should be anchored to its associated transmission element. After you click the "OK" button, the Line Flow Gauge Options Dialog will close, and the new line flow gauge will appear.

Once a gauge has been placed on the oneline, the height of its filled region will change as the flow of its transmission element changes. To modify any of the characteristics of the gauge, such as its key flow levels, fill color, and anchor setting, simply right-click on the line flow gauge to open the Line Flow Gauge Options Dialog again.

Line Flow Gauge Options Dialog

The Line Flow Gauge Options Dialog is used to define and configure a [Line Flow gauge](#). A line flow gauge is associated with a particular line and reveals the line's flow relative to specified minimum and maximum thermal flow levels. The height of the colored column in the line flow gauge indicates the line's flow relative to these markings.

The dialog has the following controls:

Number, Name, and Find

Use the **Number** and **Name** dropdown boxes, in addition to the **Circuit** field, to identify the terminal buses to which the branch desired corresponds. It may be more convenient to press the **Find** button to open the [Find Dialog](#), which allows you to specify the line by either bus names or numbers using wildcards. When you first open the Line Flow Gauge Dialog, the bus names and numbers will correspond to the transmission line object that was closest to the point where you clicked.

Minimum and Maximum

Use these two spin edit boxes in the **MW Rating values** group box to specify the minimum and maximum flow levels. These settings determine where on the gauge its two markings will be drawn.

The Minimum and Maximum value will be taken from the Limit Monitoring settings for the current transmission line, unless you uncheck the option "*Set Limits According to Current Limit Monitoring Settings*".

Anchored

A line flow gauge is said to be [anchored](#) if, when you move its associated line object, the gauge moves with it. Check the Anchored check box to ensure that the gauge will move with its associated line. Otherwise, when you move its associated line object, the gauge will stay in its current position.

OK, Help, and Cancel

Click OK to finalize your settings. This will create a new line flow gauge object if you are trying to create one from scratch, or it will modify the appearance and settings of an existing one if you have chosen to modify one that has already been defined. Click Cancel to dispose of your changes.

Line Flow Arrows on Onelines

There is a separate object, called a Line Flow Arrow, that can be added to a transmission line or transformer. This object is different than the animated flow arrows that can be added to all transmission branches through settings with the [Oneline Display Options](#). The Line Flow Arrow is a single static arrow that indicates the direction of MW flow on a transmission branch. The direction that the arrow points indicates the direction of MW flow, and there are additional options that allow the MW, Mvar, and MVA flow values to be displayed with the object.

To add a Line Flow Arrow to a transmission branch, select **Indication > Line Flow Arrow** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab while in Edit mode, then left-click on the branch to which the Line Flow Arrow should be added. The Line/Transformer Flow Object dialog will open.

The dialog contains the following options:

FromBus, ToBus, Circuit

The from bus, to bus, and circuit of the branch the arrow represents. The from bus indicates the side of the branch at which the flow will be reported. The **Find** button can be used to open a dialog that allows the selection of a branch.

Size

Size of the object.

Angle

Angle indicating how the object will be drawn. Enter this in degrees.

Anchored

Check this box to anchor the flow arrow to the branch that it represents. If anchored, the flow arrow will move accordingly any time that the branch is moved.

Auto-determine angle when anchored

Check this box to have the angle of the flow arrow determined by the branch to which it is anchored rather than using the user-specified value. The angle is determined by how the branch is drawn and the direction of MW flow on the branch. It is a good idea to use this option by default. This will ensure that the arrow is drawn correctly with the direction of flow.

Show MW, Show Mvar, Show MVA, Show Units

Check the appropriate box to show the selected flow value in text along with the flow arrow object. Check the Show Units box to include units along with the value.

Flow Text Position

Select to place any flow text either **Above Arrow** or **Below Arrow**.

Total Digits in Field, Digits to Right of Decimal

Use these fields to indicate the total digits and digits to the right of the decimal for any values that are shown in the flow text.

By default, line flow arrows will be drawn with a lime color. The color and other attributes can be changed while in edit mode. To change how the line flow arrow is drawn, select the object to change and then select **Format** from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This will open the Format Multiple Objects dialog and any attributes that can be modified will be enabled on this dialog.

DC Transmission Line Display Objects

DC transmission lines are represented on the onelines using multiple segment lines drawn between two buses. [Line fields](#) are often placed close to dc transmission lines on the oneline to indicate the power flow through the device.

Note that, unlike ac transmission lines, dc transmission lines cannot be equipped with circuit breakers.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the dc transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a dc transmission line displays the line's local menu, from which you can choose to inspect the [DC Transmission Line Dialog](#).

Edit Mode

To add a new dc transmission line to the case, first select **Network > DC Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the bus you desire to be the rectifier bus for the line and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete the new line, place the cursor on the second bus for the line, which will serve as the inverter bus, and double-click the left mouse button. This calls up the [DC Transmission Line Dialog](#). If you successfully selected the rectifier and inverter buses, their numbers will be automatically filled in for you when the dialog opens.

The DC Transmission Line Record Dialog has four separate pages: Line Parameters, Rectifier Parameters, Inverter Parameters, and Actual Flows. The separate pages can be accessed using the tabs shown at the top of the dialog. These pages are used to set the modeling parameters associated with the dc lines.

To modify the parameters for an existing dc transmission line, position the cursor anywhere on the line and right-click. This will provide access to the corresponding [DC Transmission Line Dialog](#), from which you can modify any of the dc line's parameters. Select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the color and/or line thickness of the dc line.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex by clicking and holding the left mouse button down, dragging the vertex to a new location, and releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

Multi-section Line Display Objects

Multi-section lines are represented on the onelines using a line segment with intermediate or "dummy" bus representations. These objects are not the same as a normal transmission line. They do display animated flows in run mode, but do not have pie charts, circuit breakers, or text fields currently associated with the objects.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a multi-section line displays the [Multi-section Line Information Dialog](#).

Edit Mode

Unlike transmission lines and transformers, multi-section lines CANNOT be inserted in a case by adding the multi-section line graphically. Therefore, when inserting a multi-section line object, the data for the object must already exist in the case. A list of currently defined multi-section lines is found in the model explorer under **Aggregations > Multi-Section Lines**. Right-clicking and selecting **Insert...** in the table that opens will allow you to add new multi-section line information via the [Multi-Section Line Information](#) dialog.

To insert the multi-section line object, first select **Network > MS Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transmission line (the "from" bus) and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete adding a new line, place the cursor on the second bus for the line (the "to" bus) and double-click with the left mouse button. Completing the multi-section line opens the [Multi-section Line Information](#) dialog. At the bottom of this dialog are display options for the line, such as line pixel thickness, symbol size (for the series capacitor representations, if any,) anchored, and options for drawing the buses.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex clicking and holding the left mouse button down on the vertex, dragging it to a new location, and then releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

Transformer Display Objects

Transformers are represented as transmission lines with two opposing coils drawn on one of the segments. The transformer's line thickness, color, and symbol segment can be customized using the Edit Mode. Optionally, [circuit breakers](#) and [pie charts](#) can be placed on the transformer. Clicking on the circuit breakers changes the status of the transformer.

[Line fields](#) are often placed close to transformers on the oneline to indicate the power flow through the device. [Transformer fields](#) are often placed close to transformers on the oneline to indicate and control their tap positions. See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow through the transmission line, either in MW or in terms of the transformer's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a transformer displays the line's local menu, from which you can choose to inspect the [Line/Transformer Dialog](#).

Edit Mode

New transformers are inserted in much the same way as transmission lines. To add a new transformer to the case, first select **Network > Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transformer (the *from bus*) and click the left mouse button. Add more segments to the transformer by moving the cursor and clicking with the left mouse button. To complete the new transformer, place the cursor on the transformer's other terminal (the *to bus*) and double-click with the left mouse button. This calls up the [Line/Transformer Dialog](#). The *from* and *to* bus numbers for the transformer should have been set automatically. If there is just one transformer between the buses, the circuit number should be "1." For multiple transformers between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the transformer, the number of the line segment in which you would like the transformer symbol drawn, and the size of the transformer symbol.

Enter the per unit (100 MVA base) resistance, reactance and charging susceptance for the transformer, and an MVA rating. Enter the off-nominal tap ratio and the phase shift angle in degrees. (For a transformer without tap or phase control, the off-nominal tap should be 1.0 and the phase shift angle should be 0 degrees.)

Select the appropriate Automatic Control Option. If the transformer does not have tap control, select *No Automatic Control* (this is the default). Select *AVR* (Automatic Voltage Regulation) if the transformer changes its tap ratio to control the voltage at user specified regulation bus. Select *Reactive Power Control* if the transformer changes its tap ratio to control the reactive power through the transformer. Finally, select *Phase Shift Control* if the transformer changes its phase shift to control the MW flow through the transformer. If you need any of the last three options, select the Automatic Control Options button to set the parameters associated with the automatic control.

For AVR control, enter the number of the bus whose voltage is to be controlled, the allowable range for the controlled voltage (in per unit), the minimum and maximum tap ratios (typical values are 0.9 and 1.1), and the step size for the discrete changes in the tap ratio (typical value is 0.00625).

For reactive power control, the control variable is always the reactive power measured at the *from bus* (i.e., the tapped side) of the transformer. Positive flow is assumed to be going through the transformer to the *to bus*. Enter the minimum and maximum allowable flows, the minimum and maximum tap ratios (typical values are 0.9 and 1.1), and the step size for the discrete changes in the tap ratio (typical value is 0.00625).

For phase shift control, the MW flow through the transformer is the controlled value. Enter the bus number of the terminal whose flow is controlled, the allowable range for the controlled flow (positive flow is assumed to be into the transformer at the terminal entered in the previous field), the minimum and maximum phase angles (typical values are -30° and 30°), and the step size in degrees (typical values are between 1° and 2°).

Select **OK** to save the values and return to the Transformer Options Dialog; otherwise select **Cancel**.

If you would like the transformer to be initially modeled as being on automatic control at the start of the case, select the *Automatic Control Active* checkbox.

If you do not want to add the transformer to the case, select Cancel.

To modify the parameters for an existing transformer, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the [Line/Transformer Dialog](#). Use the Line/Transformer Dialog to adjust many of the transformer's electrical properties. The [Formatting](#) ribbon group provides tools to change the transformer's color and/or line thickness.

See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

Transformer Fields on Onelines

Transformer fields are used to show field values specific to transformers, such as tap position, phase angle, and more.

Run Mode

If there is a spin control integrated with the field, you can click on the spinner to change the field's value by the associated *Delta Per Mouse Click*.

Right clicking on the transformer field gives the option to display the [Transformer Field Dialog](#) or the [Line/Transformer Information Dialog](#).

Edit Mode

To enter a new transformer field, select **Field > Transformer Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on the transformer object to which you want to add the new field. This calls up the [Transformer Field Dialog](#). Enter the *from* and *to* bus numbers associated with the device (the default values for these fields correspond to the transformer on which you clicked), and the circuit number of the device. Enter the total number of digits that the field should display, as well as the number of digits to the right of the decimal point. Finally, specify what the field should display: the off-nominal tap ratio, the off-nominal tap position, or the phase shift angle in degrees.

To modify the parameters of an existing transformer field, position the cursor anywhere on the object and right-click to bring up the [Transformer Field Dialog](#). Use the tools of the [Formatting](#) ribbon group to change various display attributes for the transformer field, including its font and background color.

Three Winding Transformer Display Objects

Three winding transformer display objects do not currently have as much functionality as other display objects. They do provide a graphical representation for identifying where three winding transformers are located in the network, but the display objects do not provide access to additional control information for the transformer. Three winding transformer display objects cannot be anchored to buses like other objects can, meaning that they will not move as other objects in the display are moved around.

A three winding transformer can be added to a oneline display while in Edit mode by selecting **Network > Three-Winding Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will change the mouse cursor to a cross-hair. Place the cross-hair on the oneline background at the location where the new transformer should be placed and then left-click. This will open the Three-Winding Transformer dialog.

The Three-Winding Transformer dialog contains information about the display of the three winding transformer only. Use the [Three Winding Transformer Display](#) to view and change control parameters and other information about the transformer. The dialog contains the following parameters:

Primary Number, Name, and Area

Number, name, and area number of the primary winding bus.

Secondary Number, Name, and Area

Number, name, and area number of the secondary winding bus.

Tertiary Number, Name, and Area

Number, name, and area number of the tertiary winding bus.

Circuit

Circuit ID of the three winding transformer.

When creating a new three winding transformer on the display, the primary, secondary, and tertiary bus numbers and circuit ID must be provided that correspond to a three winding transformer that already exists in the power flow case data model. The creation of a new three winding transformer in the data model is not allowed by graphically inserting a transformer. The bus names and area numbers are provided for informational purposes only and do not have to be entered when creating a new display object.

Click the **Find** button next to the winding specifications to open a dialog that can be used to search for an existing three winding transformer in the data model.

An existing three winding transformer display object can be assigned to a new three winding transformer data object by changing the Primary, Secondary, Tertiary, and Circuit to match an existing three winding transformer data object.

Labels

Optionally, create a label to identify the three winding transformer that the display object represents. Assigning a label here will associate a label with three winding transformer in the data model. See the [Labels](#) topic for more information about labels.

This entry is an exception for this dialog. All other fields are associated with the display object only. This is the only field that is associated with the underlying data object that the display object represents.

Display Size

This is the vertical size of the object when displayed using Orientation Up or Down. This is the horizontal size of the object when displayed using Orientation Left or Right. This is in Simulator units.

Display Width

This is the vertical size of the object when displayed using Orientation Left or Right. This is the horizontal size of the object when displayed using Orientation Up or Down. This is in Simulator units.

Pixel Thickness

Thickness in pixels of the lines used to draw the object.

Anchored

This option is permanently disabled. Currently, three winding transformer objects cannot be anchored to other objects.

Orientation

Options are Right, Left, Up, and Down. The option selected determines the side of the object that the secondary and tertiary windings are on relative to the primary winding.

Tertiary Orientation

This option determines the placing of the tertiary winding relative to the secondary winding. If the **Orientation** is either Right or Left, the Tertiary Orientation options are either Up or Down. If the **Orientation** is either Up or Down, the Tertiary Orientation options are either Right or Left.

Colors

Different colors can be specified for the Primary, Secondary, and Tertiary windings. Either left-click on the color or click the **Change** button next to the appropriate winding to open a dialog that will allow color selection.

OK, Cancel, Help

Click **OK** to apply any changes and close the dialog. Click **Cancel** to close the dialog without applying any changes. Click **Help** to open this help topic.

Series Capacitor Display Objects

Series capacitors are represented as transmission lines with two opposing parallel bars drawn on one of the segments. The series capacitor's line thickness, color and symbol segment can be customized using the Edit Mode. If the series capacitor branch is operating, but the series capacitor status is set to **Bypassed**, a low impedance segment will be drawn around the series capacitor symbol to indicate the capacitor has been bypassed. The capacitor status of **Bypassed** or **In Service** can be toggled in run mode if the Series Capacitor Status field is displayed on the oneline diagram. When a left-click is registered on the Series Capacitor Status field when in Run Mode, the capacitor status is toggled. Note that this is not the same as the overall branch status of **Open** or **Closed**.

Optionally, [circuit breakers](#) and [pie charts](#) can be placed on the series capacitor. Clicking the circuit breakers changes the branch status of the series capacitor.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow through the series capacitor, either in MW or in terms of the transformer's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a transformer displays the line's local menu, from which you can choose to inspect the [Line/Transformer Dialog](#).

Edit Mode

New series capacitors are inserted in much the same way as transmission lines. To add a new series capacitor to the case, select **Network > Series Capacitor** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the series capacitor (the *from bus*) and click the left mouse button. Add more segments to the series capacitor by moving the cursor and clicking with the left mouse button. To complete the new series capacitor, place the cursor on the series capacitor's other terminal (the *to bus*) and double-click with the left mouse button. This calls up the [Line/Transformer Dialog](#). The *from* and *to* bus numbers for the series capacitor should have been set automatically. If there is just one branch between the buses, the circuit number should be "1." For multiple branches between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the series capacitor, the number of the line segment in which you would like the series capacitor symbol drawn, and the size of the series capacitor symbol.

Enter the per unit (100 MVA base) resistance, reactance and charging susceptance for the series capacitor, and an MVA rating. On the Series Capacitor tab, check the box labeled **Is Series Capacitor** to indicate that the branch model is a series capacitor device.

If you do not want to add the series capacitor to the case, select Cancel. Otherwise click **OK** to add the series capacitor to the case.

To modify the parameters for an existing series capacitor, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the [Line/Transformer Dialog](#). Use the Line/Transformer Dialog to adjust many of the series capacitor's electrical properties.

See [Series Capacitor Information](#) for more details on modeling series capacitors.

Series Capacitor Fields on Onelines

Series capacitor fields are used to show field values specific to series capacitors.

Run Mode

If the Series Capacitor field for capacitor status is displayed, the capacitor status can be toggled in run mode when you left-click on the field.

Right clicking on the series capacitor field gives the option to display the [Series Capacitor Field Dialog](#) or the [Line/Transformer Information dialog](#).

Edit Mode

To enter a new series capacitor field, select **Field > Series Capacitor Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on or near the series capacitor object for which you want to add the new field. This calls up the [Series Capacitor Field Dialog](#). Enter the *from* and *to* bus numbers associated with the device (the default values for these fields correspond to the series capacitor on which you clicked), and the circuit number of the device. Enter the total number of digits that the field should display, as well as the number of digits to the right of the decimal point. Finally, specify what the field should display: the capacitor status or the series capacitance.

To modify the parameters of an existing series capacitor field, position the cursor anywhere on the object and right-click to bring up the [Series Capacitor Field Dialog](#).

Switched Shunt Display Objects

Switched shunts are either capacitors that supply reactive power to the system or reactors that absorb reactive power. Simulator represents switched shunts as a number of blocks of admittance that can be switched in a number of discrete steps or over a continuous range. Switched shunt display objects come equipped with a circuit breaker that indicates the shunt's status. If the switched shunt is closed, the circuit breaker appears as a filled red square. If the switched shunt is open, the circuit breaker appears as a green square outline. To change the status of the switched shunt, click the corresponding circuit breaker.

[Switched shunt fields](#) are often placed next to switched shunts to indicate the amount of reactive power supplied by the device. For switched shunts with such a field, you can manually increase the reactive power supplied by the device (provided its control mode is discrete) by clicking on the up-arrow associated with the device's reactive power field. Likewise, you can decrease the reactive power supplied by the device by clicking on the down-arrow. To make the up/down arrows visible, set the *Delta per Mouse Click* on the [switched shunt field](#) to a nonzero value.

Right-clicking on the switched shunt displays the [Switched Shunt Dialog](#). Use the Switched Shunt Dialog to inspect or modify the model of the switched shunt.

You can add a new switched shunt to the case in Edit Mode. Select **Network > Switched Shunt** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then click the bus where you would like to attach the device. The [Switched Shunt Dialog](#) will appear. The bus number is automatically determined from the bus to which you attached the capacitor. Enter the size, the thickness of the pen [in pixels] used to draw the device, and its orientation. The Nominal Mvar field gives the amount of reactive power the device would supply if its terminal voltage were 1.0 per unit. The Control Mode field determines whether the switched shunt has a fixed value or will vary discretely or continuously within its operating limits to maintain its terminal voltage within the voltage range specified in the Voltage Regulation field.

The amount of shunt admittance is specified in the Switched Shunt Blocks table. The columns in this field correspond to different blocks of admittance. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step. The switched shunts are always switched in the order specified in this field.

Select OK to add the device. If you do not want to add the switched shunt to the case, select Cancel.

To modify the parameters for an existing switched shunt, position the cursor on the device and right-click. This again brings up the [Switched Shunt Dialog](#). You can then change any parameter of the switched shunt. Make use of the tools in the [Formatting](#) ribbon group to change the color and/or line thickness.

To delete an existing switched shunt, use the **Cut** or **Delete** commands in the [Clipboard](#) ribbon group.

Switched Shunt Fields on Onelines

Switched Shunt field objects are used primarily to indicate various quantities associated with switched shunt devices. Furthermore, some switched shunt field types, which are distinguished by an integrated spin button, may be used to change switched shunt device properties.

Run Mode

For switched shunt fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a generator field gives you the option to open the [Switched Shunt Field Dialog](#) or the [Switched Shunt Information Dialog](#).

Edit Mode

Simulator offers two options for adding switched shunt fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Switched Shunt Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then select the switched shunt to which you want to add the field. This invokes the [Switched Shunt Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest switched shunt to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show.

The second approach for adding new switched shunt fields entails right-clicking the bus and selecting *Add New Fields Around Switched Shunt* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing switched shunt field, position the cursor anywhere on the object and right-click. This brings up the [Switched Shunt Field Dialog](#). Make use of the tools on the [Formatting](#) ribbon group to change various display attributes of the field, including its font and background color.

Interface Display Objects

Interface display objects are used on the onelines to visualize the flow of power through an [interface record](#). Interface records are used to show the net real power (MW) flow on a group consisting of one or more of the following devices: 1) transmission lines and/or transformers, 2) total tie-lines between two adjacent areas, and 3) total tie-lines between two adjacent zones. Only area-area and zone-zone interface records can be displayed using interface display objects.

Interfaces, like transmission lines, are represented as multi-segment lines, except that they may be drawn between [area/zone objects](#) in addition to buses. Drawing interface display objects involves the same steps as drawing transmission lines and transformers. The line thickness and color of interface objects may be customized by selecting  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

[Interface Fields](#) and [Interface Pie Charts](#) are often placed close to or on the interface to indicate the power flow through the device.

Run Mode

When animation is active, the flow of the arrows on the interface object may represent either the MW flow through the interface or the currently calculated power transfer distribution factor ([PTDF](#)) pertaining to that interface. You can customize the appearance of the animated flows using the *Animated Flows Tab* of the [Oneline Display Options Dialog](#) available using the local menu.

Right-click on the interface to view the [Interface Dialog](#) for the interface.

Edit Mode

The quickest method of inserting new interface objects is to use the **Auto Insert > Interfaces** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. Please see [Automatically Inserting Interfaces](#) for details.

To manually add a new interface object to the case, first select **Aggregation > Interface** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the starting location for the interface (usually an area/zone object or a bus object) and left-click. Add more segments to the interface by moving the cursor and the left-clicking at the end of the segment. To finish adding an interface, place the cursor on the terminal object for the interface and double-click. This then calls up the [Interface Dialog](#). Either select an existing interface, or define a new interface (see [Interface Dialog](#) for details).

Interfaces can be [anchored](#) to either area/zone objects or bus objects.

To change the shape of the interface, first left-click on the object to select it. This causes handles to appear at each vertex. You can then move any vertex by dragging it with the left mouse button down. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the interface where you would like to add the vertex.

Automatically Inserting Interface Display Objects

The Automatic Insertion of Interfaces Dialog is used to automatically insert [interface objects](#) on the oneline diagram between existing [area/zone display objects](#). [Interface pie chart objects](#) can also be inserted as part of this process. The automatic insertion of interface display objects greatly accelerates the construction of interface diagrams, which are particularly useful for animating the results of [PTDF](#) calculations. Area-area and zone-zone interface records can be displayed using interface display objects as well as interfaces comprised only of transmission branches.

Note that inserting interface objects on a oneline diagram does NOT add interface objects to the load flow case. The interface definitions need to be added to the load flow case prior to automatically inserting the graphical interface objects on a oneline diagram. The quickest way to add area to area or zone to zone interface definitions in the load flow case is to [automatically insert the definitions](#) in the interface case information display.

Once you have the interface definitions defined in the case, you can automatically insert the interface objects on a diagram using the following procedure:

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **New Oneline** from the [application button](#)), place [area/ zone objects](#) at the desired locations if inserting area-area or zone-zone interface records. Otherwise, the terminal buses for the branch elements in the interface should be inserted at the desired locations on the oneline diagram.
- Go to the [Draw](#) ribbon tab and choose **Auto Insert > Interfaces** on the [Quick Insert](#) ribbon group.
- Check the **Insert Pie Charts on Interfaces** box to automatically insert interface pie chart objects when the interfaces are inserted. If this option is selected, change the **Default Size of Interface Pie Charts** to specify their size.
- Choose the type of interfaces to insert. **Area to Area or Zone to Zone Interfaces** and **Line/Transformer Interfaces** can be inserted. Area-area and zone-zone interfaces will be drawn between the respective area and zone objects displayed on the oneline. Line/transformer interfaces will be drawn based on the average location of the terminal buses of all transmission lines comprising the interface. If inserting line/transformer interfaces, the option **Minimum Length of Line/Transformer Interfaces** can be set to prevent interfaces that are too short from being inserted.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted based on the selected criteria for any corresponding interface record that is not already represented.

Note that you can do this automatic insertion as often as you like. The **Number of Interfaces Not Shown** field indicates how many interfaces still need to be added to the diagram to represent all defined area-area, zone-zone, or line/transformer interfaces. It is a read-only field.

Interface Fields on Onelines

Interface field objects are used to show values associated with [interface records](#).

Run Mode

Right clicking on the interface field displays the [Interface Field Dialog](#).

Edit Mode

To enter a new interface field, select **Field > Interface Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on the background of the oneline diagram where you want the field placed. This calls up the [Interface Field Dialog](#). Enter the name of the interface, the total number of digits desired in the field, and the type of field.

To modify the parameters of an existing interface field, position the cursor anywhere on the object and right-click. This again brings up the Interface Field Dialog.

InterArea Flow Options Dialog

This dialog is outdated in PowerWorld Simulator version 5 and later. See help on the [Interface Field Information Dialog](#) for the updated dialog.

When viewing an area diagram containing inter-area objects created using PowerWorld Simulator version 4.2 or older, this dialog allows you to set text fields displaying either the actual or scheduled MW flow on the inter-area object.

Interface Pie Charts on Onelines

Interface pie charts are used to graphically show the percentage loading on an [interface record](#). The amount of shaded region of the pie chart indicates how close the interface is to its limit (provided the interface has a nonzero limit). The appearance of the interface pie charts, including their color and the ability to automatically change size based upon loading level, can be customized on the [Online Display Options Dialog](#).

Right-clicking on the interface pie chart displays the [Interface Pie Chart Dialog](#). This dialog can be used to customize the size of the pie chart, or change the interface's limit.

Edit Mode

To enter a new interface pie chart, select **Pies/Gauges > Interface Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse on the interface object with which you want to associate the pie chart. This calls up the [Interface Pie Chart Dialog](#), which is used to customize the appearance of the pie chart.

Loading NERC Flowgates

This command reads flowgates from an Excel file and inserts them as interface records. The format for this file should be similar to the files found at <http://www.nerc.com/~oc/dfwg.html>. To access this file you must have a NERC-supplied username and password. At the time of writing we have not seen an official description of the format -- currently Simulator just mimics the format found in this file. Flowgates are used by NERC (under Policy 9) as proxies for transmission limitations and transmission service usage on the interconnected electric power network. Simulator models flowgates using the interface records. Interface records MAY include contingency elements.

The **Load NERC Book of Flowgates from Excel** option is available from the [local menu](#) of the [Interface Records](#) display.

Saving NERC Flowgates

This command writes all the interfaces to an Excel spreadsheet using the NERC flowgate format. See the files found at <http://www.nerc.com/~oc/dfwg.html> for an example of the NERC format. To access this file you must have a NERC-supplied username and password.

The **Send NERC Book of Flowgates to Excel** option is available from the **Save As** option of the [local menu](#) of the [Interface Records](#) display.

Injection Group Display Objects

Injection group display objects are used on onelines to show a summary of the total injection of an [injection group](#) from generation and load. The summary shows only the MW injection.

An injection group display object can be added to a oneline by choosing **Aggregation > Injection Group** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. After selecting this option, place the cursor at the location on the oneline where the injection group object should be placed and left-click. This will open the Injection Group Display Options dialog with the following options available:

Name

Select the name of the injection group represented by the current display object. The drop-down will contain the names of all currently defined injection groups. New injection groups cannot be created through the injection group display object.

Style

The style dictates how the object will be drawn on the oneline. The injection group object is a simple object that is either a Rectangle or Rounded Rectangle.

Caption

This option is used to determine how the injection group will be identified on the display object. Currently, only the Name can be used as the identifier.

Width, Height

Specify the size of the object through these two options.

After an object has been created, the Injection Group Display Options dialog can be opened while in Edit mode by right-clicking on the object. If in Run mode, the [Injection Groups Dialog](#) will open when right-clicking on an injection group display object.

An injection group display object uses the same [Default Drawing Values](#) as those specified for an area display object. To change how the object is drawn after it has been created, select the object to change and then select **Format** from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This will open the Format Multiple Objects dialog, and any attributes that can be modified will be enabled on this dialog. These format changes can only be done while in Edit mode.

Links to Onelines and Auxiliary Files

Run Mode

Online Links

Online links are one of the mechanisms used in Simulator to allow you to view multiple oneline diagrams on the same screen. By default, the oneline links are shown as blue rectangles surrounding a text identifier of the linked oneline diagram. Double-clicking the left mouse button anywhere within the oneline link object will display the linked oneline diagram, even if it has not already been opened.

The ability to associate multiple oneline diagrams with a single case may prove particularly helpful when dealing with large cases. If a single oneline diagram is used to display a case having many buses, the diagram may become cluttered, and it will be difficult to analyze the case in sufficient detail. The ability to associate additional onelines with the case and to call up those additional onelines using one-links can significantly enhance your view of the system.

Online Links can be used to open oneline diagrams saved in either the PowerWorld Display file (*.pwd) or Display Auxiliary file (*.axd).

Note that you can also use the **Open Oneline** command from the [Application Button](#) to open any oneline diagram (*.pwd, *.axd) file directly. See [Opening a Oneline Diagram](#) for further details.

Auxiliary File Links

Auxiliary (*.aux) file links are one of the mechanisms used in Simulator to allow you to load auxiliary files by simply clicking on a oneline object. By default, auxiliary file links are shown as blue rectangles surrounding a text identifier of the linked auxiliary file. Double-clicking the left mouse button anywhere within the auxiliary file link object will load the associated auxiliary file.

Note that you can also use the **Load Auxiliary File** command from the [Application Button](#) to open any auxiliary file (*.aux) directly. See [Auxiliary Files](#) for further details.

Edit Mode

Online Links

To add a new oneline link to a oneline diagram, first select **Background > Oneline Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse button at the location where you would like to display the new link. The Oneline Link Options Dialog box will open, asking you to enter the name of the file that stores the linked oneline and a caption for the clickable oneline link object. The **Browse** button can also be used to open a dialog that will allow searching for the file. Note that you can enter simply the name of the oneline file in this location or the complete directory path and name. Simulator will look in the same directory as the case file by default. However, you can also specify additional directory locations to search for oneline diagrams by opening the [Oneline Display Options Dialog](#) (via **Oneline Display Option** from the **Oneline Options** ribbon group on the [Options](#) ribbon tab, or from the [Active](#) ribbon group on the [Onelines](#) ribbon tab), switching to **Display Options** on the left, and using the **Edit Oneline Browsing Path** option to add additional search locations for oneline diagrams.

Oneline diagrams stored in either the PowerWorld Display format (*.pwd) or Display Auxiliary format (*.axd) can be opened using oneline links. If using a Display Auxiliary format (*.axd) file, the browsing path will not be used to find the file. The file must exist in the current directory or the full directory path of the file must be specified.

Auxiliary File Links

To add a new auxiliary file link to a oneline diagram, first select **Background > Auxiliary File Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse button at

the location where you would like to display the new link. The Online and Auxiliary Link Options Dialog will open, asking you to enter the name of the file that stores the linked auxiliary file and a caption for the clickable auxiliary file link object. Note that you can simply enter the name of the auxiliary file in this location. Simulator will look in the same directory as the case file.

Modifying Existing Links

To modify the parameters of an existing online or auxiliary file link, position the cursor anywhere on the object and right-click. This invokes the Online Link Options Dialog, allowing you to change the identity of the linked file and the caption. Select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to modify various display attributes of the online link, including font size and background color.

Document Links on Onelines

Just as you can link to other oneline displays from a oneline diagram using [oneline link objects](#), you can also link to documents and data on the world-wide web using Document Link Objects. When you click on Document Link Objects in Run Mode, your system's default browser will be launched to retrieve the linked URL address. This feature is not only for web URL's, though. Any file can be linked and its associated application will automatically open. This means that presentations, documents and spreadsheets can also be linked.

In addition to linking to objects online via a URL, it's also possible to put in a local file name and have that open when the document link is clicked. The behavior is the same as if you were to click on the Windows Start button, then select Run..., then type in the file name.

To add a Document link to a oneline diagram, select **Background > Document Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse on the oneline diagram at the position where you would link to insert the Document link. The Document Link Options Dialog box will appear, asking you to specify the world-wide web address to which to link as well as a clickable caption to display on the oneline diagram. Enter the requested information and press *OK* to add the Document link object to the oneline.

To modify the caption or address for a Document link object, right-click anywhere on its text. Specify its new parameters in the Document Link Options Dialog and press *OK*. You may also modify various aspects of its appearance, including the font size, by choosing  from the [Formatting](#) ribbon group.

Note that Document links are active only in Run Mode. Clicking on a Document link object from the Edit Mode will have no effect other than to select the object for placement, formatting, etc.

Background Lines on Onelines

The background of an oneline diagram can display added lines, polylines, and filled polygons among other items.

Edit Mode

To add a new background line, first go to the [Draw](#) ribbon tab and select **Background > Background Line** from the [Individual Insert](#) ribbon group. Then to draw a series of straight line segments on the background, follow these steps:

- Position the mouse cursor where you want the series to begin and click and release the left mouse button.
- Move the mouse to the desired termination point of the first line segment. A straight segment will follow your mouse movements. Click and release the left mouse button to complete the line segment and prepare for drawing the next line segment, or double-click if this line segment is the last segment you wish to draw.

To draw a freehand shape rather than a series of straight line segments, click and hold the left mouse button where you would like the freehand shape to begin and drag the mouse to trace the shape you desire (while holding the left mouse button down). Release the left mouse button to complete the section of the freehand shape you have been drawing. At this point, you can add either another freehand section or a straight line segment. When you have finished drawing in the background, double click the mouse button.

Note that background display objects composed of straight line segments display significantly faster than lines drawn freehand. Lines drawn freehand (holding down mouse button) leave a vertex point at every point on the screen, where a line composed of straight-line segments (left-clicking only where you want a vertex point) takes considerably less effort for the PC to draw.

To change the color, line thickness, and fill color associated with the line, make use of the tools in the [Formatting](#) ribbon group.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex by holding the left mouse button down and dragging the vertex to a new location. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex. Note that freehand lines are nothing more than a continuous series of vertices.

Background Lines Dialog

The Line Options Dialog is available while in edit mode by right-clicking on a background line and selecting **Open Dialog...** This dialog contains options for formatting the selected background line. Many of the options listed in this dialog can also be updated on the [Format](#) dialog.

Line Thickness

This field gives the thickness of the line in pixels.

Line Color

This field shows the current line color. To change the line's color, click on the box displaying the current color or click the **Change** button next to the line color. This will display the Color Dialog. Select the desired color and select OK to change the color.

Use Background Fill

When this option is checked, the selected Fill Color will be used to fill the background of the line.

Fill Color

This field shows the current fill color. To change the fill color, click on the box that displays the current fill color or click the **Change** button. This will display the Color Dialog. Select the desired color and select OK to change the color.

Immobile

When this option is checked, the background line will be forced to stay in the same position and cannot be moved by dragging it with the mouse.

List of Vertices

This grid lists the x,y coordinates for the vertices of the line in the order in which the line is drawn. The coordinates will be displayed as longitude,latitude if the [option to show coordinates in longitude,latitude](#) is selected on the [Online Display Options](#) dialog and a valid map projection is in use. The values for the coordinates can be changed by entering new values in the appropriate position in the grid.

Shift X/Y Values

These options provide a means of updating all of the x and/or y values. The **X Shift Value** is the offset by which all of the x values will be shifted. The **Y Shift Value** is the offset by which all of the y values will be shifted. The resulting values will be the original values plus the value entered for the shift. The **Shift All Values** button must be selected for the values entered in the shift fields to be applied to the x,y values. After this button is selected, the coordinates shown in the grid will be updated with the new values.

Scale X/Y Values

These options provide a means of updating all of the x and/or y values. The **X Scale Value** is the value by which all of the x values will be scaled. The **Y Scale Value** is the values by which all of the y values will be scaled. The resulting values will be the original values multiplied by the value entered for the scale. The **Scale All Values** button must be selected for the values entered in the scale fields to be applied to the x,y values. After this button is selected, the coordinates shown in the grid will be updated with the new values.

OK, Save and Update Display, and Cancel

OK saves the changes and closes the dialog. **Save and Update Display** saves the changes and updates the display without closing the dialog. **Cancel** closes the dialog without saving any changes.

Background Rectangles on Onelines

The background of a oneline diagram can display rectangles among other items.

Edit Mode

To add a new rectangle, first go to the [Draw](#) ribbon tab and select **Insert > Background Rectangle** from the [Individual Insert](#) ribbon group. Position the cursor where you would like to place the upper left-hand corner of the rectangle and click with the left mouse button. A rectangle having the default size is inserted. Drag the rectangle's resizing handles to resize/reshape the rectangle.

To resize or reshape an existing rectangle, click on it to select it. The resizing handles will appear, which you can then drag to reshape or resize the rectangle.

To change the color, line thickness or fill color of the rectangle, first select the rectangle by clicking on it on the diagram, and then use the tools in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab, or right-click on it and select **Format Background Rectangle** from the local menu.

Background Ellipses on Onelines

The background of a online diagram can display ellipses among other items.

Edit Mode

To add a new ellipse, first go to the [Draw](#) ribbon tab and select **Background > Background Ellipse** from the [Individual Insert](#) ribbon group. Then position the cursor where you would like to place the upper left-hand corner of the ellipse and click with the left mouse button. An ellipse having the default size and shape is inserted. Drag the ellipse's resizing handles to resize/reshape it.

To resize or reshape an existing ellipse, click on it to select it. The resizing handles will appear, which you can then drag to reshape or resize the ellipse.

Right-click on the ellipse and choose **Format Background Ellipse**, or select the ellipse and use the tools in the [Formatting](#) ribbon group found on the [Draw](#) ribbon tab to change the color, line thickness and fill color of the ellipse.

Background Pictures on Onelines

The background of an oneline diagram can display a variety of objects, including lines, filled polygons, and even pictures. The latter will be discussed in this section.

Edit Mode

Simulator can insert bitmaps, jpegs, enhanced and standard metafiles, and icons from files into the oneline diagram. These pictures may either serve as a background or appear above other objects on the oneline.

To add a picture object to the oneline diagram, go to the [Draw](#) ribbon tab and select **Background > Picture** from the [Individual Insert](#) ribbon group. Then, click the oneline diagram at the point where you would like the top left corner of the graphic to be placed. The Open Picture Dialog will open, asking you to select the graphic file that you want to insert. When you select a file from the dialog, the dialog displays a preview image so that you can be sure that you are selecting the right file. When you have identified the file to insert, click *OK*. The image will then appear on the oneline. It may be resized by dragging its resizing handles.

By default, pictures are inserted at the middle stack level. Thus, they hide most other oneline display objects. To change the stack level of the picture, click it to select it, and then select the *Levels/Layers* button found in the [Formatting](#) ribbon group of the [Draw](#) ribbon tab. From the *Levels/Layers Tab*, select the stack level of your choice. The *Base* stack level places the picture below all other oneline display objects, while the *Top* stack level will cause the picture to obscure all other display objects.

To resize or reshape an existing picture object, click on it to select it and then drag its resizing handles.

Converting Background Lines

Background lines can be converted to other objects by splitting, merging, or converting to power system objects. To convert a background line, right-click on a selected background line on the oneline display and the local menu will be displayed with the conversion options. Most options are available if only a single background line has been selected and no other objects are selected. The following options are available for converting a background line:

Split Background Line

A background line can be split at a vertex or anywhere along the line. Click the point on the line where the split should be made and then right-click and select the **Split Background Line...** option from the local menu. Two background lines will result. This option is available only if exactly one background line has been selected and no other objects have been selected.

Merge Background Lines

This option is only available if exactly two background lines have been selected. The two selected background lines will be merged at the two closest ends and a single background line will result.

Convert to ac Transmission Line

After selecting this option, the [Line/Transformer Information dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting an ac transmission line from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

Converting Background Ellipses

Background ellipses can be converted to power system objects. To convert a background ellipse, right-click on a selected background ellipse on the oneline display and the local menu will be displayed with the conversion options. The following options are available for converting a background ellipse:

Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background ellipse has been selected and no other objects have been selected.

Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background ellipse has been selected and no other objects have been selected.

Converting Background Rectangles

Background rectangles can be converted to power system objects. To convert a background rectangle, right-click on a selected background line on the oneline display and the local menu will be displayed with the conversion options. The following options are available for converting a background rectangle:

Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background rectangle has been selected and no other objects have been selected.

Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background rectangle has been selected and no other objects have been selected.

Oneline Fields

The Oneline Fields are fields that can be placed on an oneline diagram that display values specific to the current display of the diagram, such as x and y coordinate of the mouse cursor, animation rate, zoom percentage, and x and y coordinate of the center of the diagram at the center of the screen. These fields can be inserted on a oneline diagram by choosing **Field > Oneline Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Text on Onelines

Text display objects are used to show single lines of text on the oneline.

Edit Mode

To add descriptive text to the oneline, select **Background > Text** in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and click the oneline where you would like the text to appear. The Text Options Dialog will open, asking you to enter the desired text string, and the angle at which the text is to appear on the oneline diagram.

To modify an existing text object, position the cursor anywhere on the text and right-click. The New Text Options Dialog will appear, allowing you to edit the text. Use the font option  of the [Formatting](#) ribbon group to control the font and background color of the text object.

Memo Text

Memo Text display objects are used to show several lines of text on the oneline within an enclosing box.

Edit Mode

To add a Memo Text object to the oneline, select **Background > Memo Text** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then click and drag to create a rectangle on the oneline; this rectangle will contain the lines of text you enter.

To modify the contents of a Memo Text object, position the cursor anywhere on the object and right-click. To change the formatting of a Memo Text object, select the object with a left-click, then select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab; this will bring up the [Font Properties](#) tab of the [Format Multiple Objects](#) dialog box.

Generic Model Fields

The Generic Model Fields are fields that can be placed on an online diagram that display any value for any object in the case on the diagram. These fields can be inserted on an online diagram by choosing **Field > Generic Model Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

This type of field comes in handy when you want to place a variety of fields on the diagram, without inserting specific object-type fields. All fields of data are available in one location for placement on a diagram.

Supplemental Data Fields on Onelines

Supplemental Data Field objects represent information about [Supplemental Data](#) records. The information that can be represented in these fields is the same in either Edit or Run Mode.

Supplemental Data Fields can be added to a oneline while in Edit Mode from **Draw > Field > Supplemental Data Field**. After selecting this option, move the cursor to the location on the oneline where the new field should be placed and left-click. The Supplemental Data Field Options dialog will open.

The **Supplemental Data Field Options** dialog can be used to modify the properties of individual supplemental data fields on the diagram. This dialog contains the following fields:

Classification

Select the Supplemental Classification from the drop-down list of available classifications.

Name

Select the Name of the Supplemental Data record from the drop-down list of available names. This list is populated with the appropriate names based on the selected Classification.

Total Digits in Field

Total number of digits to show in the field.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Rotation Angle in Degrees

The angle at which the text will be displayed on the diagram.

Type of Field

Select a field from the drop-down list of available fields or click **Find Field** to search through the list of available fields.

Select **OK** to close the dialog and save the changes, or click **Cancel** to close the dialog without saving the changes.

Pie Charts/Gauges: Lines

Pie Chart / Gauge Style

The Pie Chart Style box determines whether the [line flow pie charts](#) show the percentage loading of the line based upon the MVA flow, the MW flow, the MVR flow, the line amp/transformer MVA flow, the maximum percentage loading under contingency, or the PTDF value. The Line Gauge Style box has the same setting options as the line flow pie charts.

Always Use Limiting Flow

Typically, the flow at opposite ends of a transmission line is slightly different due to losses across the line. If this option is checked, the pie chart will correspond to limiting MVA value of the line, independent of which end of the line that value occurs. Otherwise, the pie chart will always show the MVA value at the from bus for the line.

Display Gauge Values in Percent

Check this option to display values in the line gauges as percent of loading instead of actual flow values.

Color, Size and Percentage

This section of the options dialog allows you to customize the appearance of the line flow pie charts on the diagram. The default options for the line flow pie charts are set on the MVA tab, and are always in force for at least the MVA pie chart / gauge style. However, you can choose different settings for the other styles by selecting the corresponding tab under the Color, Size and Percentage section. Each of the other five styles can be set to use the same settings as defined for MVA by checking the given option on the page. If this option is unchecked for a particular style, then any pie chart on the diagram of that style type will use the specifically defined appearance options for that style type.

The following parameters are all available on each of the six style tab pages:

Show Value Percent

When a branch's loading exceeds the value specified in the **Show Value Percent** field, the percentage loading is shown as text within the pie chart. The default is 80%.

Normal Size Scalar, Normal Color

The standard, pre-warning fill color and scaling factor for pie charts.

Warning / Limit Scalars and Colors

The table in this section allows you to choose different settings for the pie chart size and color, based on the percentage value represented by the pie chart or gauge. Thus you can set up visual clues as to when the flow on lines exceeds specified warning or limit levels. You can add and remove points from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, simply click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color Chooser dialog.

Make normal color the same as the line to which the pie chart or gauge is anchored (if any)

If checked, this option will cause the pie charts to assume the same normal colors as the transmission lines to which they are anchored. This may be useful if the transmission lines are colored according to their nominal voltage level.

Only Apply Warning/Limit Colors and Resizing to Monitored Elements

If this option is checked, then only those pie charts that correspond to branches selected using the [Limit Monitoring Settings](#) will change appearance to reflect warning and limit loading levels. If this option is not checked, then all pie charts will obey the options prescribed in this dialog.

Some additional options are available for pie chart appearance in situations where the line itself is out of service. Clicking on the tab labeled **Open Parameters** will display these options. You can choose to have special formatting for open devices by checking the available option, and then defining the appearance options for the pie charts on the open lines. The available options include scaling the size, width of the border, border color, pie chart background color, and drawing an "X" symbol through the pie chart.

Pie Charts/Gauges: Interfaces

Pie Chart Style

The Pie Chart Style box determines whether the interface pie charts show the percentage loading of the line based upon the MW flow, the maximum percentage loading under contingency, or the PTDF value.

Color, Size and Percentage

This section of the options dialog allows you to customize the appearance of the interface flow pie charts on the diagram. The default options for the interface flow pie charts are set on the Lines MVA tab. However, you can choose different settings for the interface styles by selecting the corresponding tab under the Color, Size and Percentage section. Each of the styles can be set to use the same settings as defined for Lines MVA by checking the given option on the page. If this option is unchecked for a particular style, then any interface pie chart on the diagram of that style type will use the specifically defined appearance options for that style type.

The following parameters are all available on each of the six style tab pages:

Show Value Percent

When an interface's loading exceeds the value specified in the **Show Value Percent** field, the percentage loading is shown as text within the pie chart. The default is 80%.

Normal Size Scalar, Normal Color

The standard, pre-warning fill color and scaling factor for pie charts.

Warning / Limit Scalars and Colors

The table in this section allows you to choose different settings for the pie chart size and color, based on the percentage value represented by the pie chart or gauge. Thus you can set up visual clues as to when the flow on interfaces exceeds specified warning or limit levels. You can add and remove points from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, simply click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color Chooser dialog.

Make normal color the same as the line to which the pie chart or gauge is anchored (if any)

If checked, this option will cause the pie charts to assume the same normal colors as the interface to which they are anchored. This may be useful if the interfaces are colored differently on the diagram.

Only Apply Warning/Limit Colors and Resizing to Monitored Elements

If this option is checked, then only those pie charts that correspond to interfaces selected using the [Limit Monitoring Settings](#) will change appearance to reflect warning and limit loading levels. If this option is not checked, then all interface pie charts will obey the options prescribed in this dialog.

Pie Charts/Gauges: Pie Chart/Gauge Styles

This section shows the Pie Chart/Gauge styles in a case information display. The user can edit several of the fields directly. Otherwise, by right-clicking on any part of the case information display and selecting on **Show Dialog** in the popup menu, the [Pie Chart/Gauge Style dialog](#), where all the values can be modified.

Pie Charts/Gauges: General Options

Show Limit Set in Header

If this option is checked, the pie chart will display a text field showing the letter of the limit set presently being used for the element.

Show Style in Footer

When checked, the pie chart will display a text field showing the style (MVA, MW, etc) that the pie chart is set to (refer to the Style property described above.)

Max. Zoom Percentage for Full Resizing

The pie charts dynamically resize when you are zooming in and out on a oneline diagram. However, you can limit the point at which the pie charts resize when zooming in by setting a zoom level in this field. This helps prevent the pie charts from getting so large that they occupy the entire screen.

Min. Pie Chart Font Size for Warning/Limit

Specifies a minimum font size for displaying text in the pie chart object. This is useful when zooming out on the diagram, to keep the text visible by setting a minimum font size.

Background Color

The background color for the pie charts is automatically set to be the same as the online background. For a different color, select Specific color from the drop-down menu, and then click on the rectangle to the right to select the color. For a clear background, select Clear from the dropdown menu.

Pie Chart Relative Font Size

Slide the slider bar to select the font size for the pie chart. This will determine the size of the font indicating the percentage on the pie chart.

Pie Chart Font Color

Slide the slider bar to select the font size for the pie chart. This will determine the size of the font indicating the percentage on the pie chart.

Pie Chart / Gauge Dialogs

There are a number of pie chart/gauge objects that can be added to one-line diagrams. These objects are inserted via the **Pies/Gauges** menu in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and include Areas, Buses, Generators, Injection Groups, Owners, Substations, Super Areas, Switched Shunts, Transformers, and Zones. The dialogs for these objects contain a common set of options that are required regardless of the object type. These common options as well as specific settings for the different object types are detailed below.

The [Pie Chart / Gauge Example](#) shows how the fields in this dialog affect the pie chart or gauge display. The items shown in the diagram are explained in more detail throughout this section and in the [Pie Chart / Gauge Style Dialog](#) description.

Identification

All dialogs have a section at the top that identify the pie chart/gauge object based on the type of device that the pie chart/gauge represents. The **Find...** button is available on most dialogs to open the Find tool that can be used to advanced search for a device. Specific identifying information for the different dialogs is given below:

- **Area**The area that the pie chart/gauge represents is identified based on **Area Number** or **Area Name**. To change the area that is represented, select an **Area Number** or **Area Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all areas in the case.
- **Bus**The bus that the pie chart/gauge represents is identified based on **Bus Number** or **Bus Name**. To change the bus that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all buses in the case with valid area/zone/owner filters.
- **Generator**The generator that the pie chart/gauge represents is identified based on **Bus Number** and **Gen ID** or **Bus Name** and **Gen ID**. To change the generator that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes and change the **Gen ID** in the edit box or click the **Find** button. The dropdown boxes are populated with all buses in the case with valid area/zone/owner filters.
- **Injection Group**The injection group that the pie chart/gauge represents is identified based on the **Injection Group Name**. To change the injection group that is represented, select a new name from the dropdown box. The dropdown box is populated with all injection groups in the case.
- **Owner**The owner that the pie chart/gauge represents is identified based on **Owner Number** or **Owner Name**. To change the owner that is represented, select an **Owner Number** or **Owner Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all owners in the case.
- **Substation**The substation that the pie chart/gauge represents is identified based on **Substation Number**, **Substation Name**, or **Substation ID**. To change the substation that is represented, select a **Substation Number**, **Substation Name**, or **Substation ID** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all substations in the case.
- **Super Area**The super area that the pie chart/gauge represents is identified based on **Super Area Name**. To change the super area that is represented, select a new name from the dropdown box or click the **Find** button. The dropdown box is populated with all super areas in the case.
- **Switched Shunt**The switched shunt that the pie chart/gauge represents is identified based on **Bus Number** and **ID** or **Bus Name** and **ID**. To change the switched shunt that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes and change the **ID** in the edit box or click the **Find** button. The dropdown boxes are populated with all switched shunts in the case with valid area/zone/owner filters.

- **Transformer**The transformer that the pie chart/gauge represents is identified based on **Near Bus Number, Far Bus Number, and Circuit** or **Near Bus Name, Far Bus Name, and Circuit**. To change the transformer that is represented, click on the Find button to use the **Find** tool.
- **Zone**The zone that the pie chart/gauge represents is identified based on **Zone Number** or **Zone Name**. To change the zone that is represented, select a **Zone Number** or **Zone Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all zones in the case.

Type of Field

Use this option to select the field to display in the pie chart/gauge. Fields that are deemed of more common interest are listed for easy access. Any field associated with a particular device can be selected by first choosing **Select a Field** and then finding the particular field in the dropdown listing all available fields or by clicking **Find Field** and searching for the field of interest.

Current Value

Percent

This is the percent value calculated from the current value of the selected field. How the percent is calculated is based on the selection of limits and deadband and options associated with the Style. Details of the percent calculation are given in the **Pie Chart/Gauge Style Dialog** description.

Value

This is the current value of the selected field.

In-Service

When checked, the device represented by the pie chart/gauge is in-service. This is given for informational purposes only, as the status of devices cannot be changed from this dialog.

Display Information

Type

Select to display either a Pie Chart or Gauge.

Gauge Orientation

If **Type** option of Gauge is selected, set this option to draw either a Vertical or Horizontal gauge.

Anchored

When checked, the pie chart/gauge will be anchored to the display object representing the selected device.

Size, Width, Use Fixed Gauge Width/Size Ratio

Size determines the size of a pie chart and the vertical side length of a vertical gauge or horizontal side length of a horizontal gauge. **Width** is not applicable to a pie chart but determines the horizontal side length of a vertical gauge or the vertical side length of a horizontal gauge. **Width** is not enterable if the option of **Use Fixed Gauge Width/Size Ratio** is checked. This option is checked if the **Gauge Width is Always a Fixed Proportion of Size** option is checked on the Style dialog. The fixed width/size ratio is also set on the **Style** dialog.

Ignore Dynamic Sizing

When checked, any sizing based on the calculated percent will be ignored and the pie chart/gauge will stay at the defined size. Options to set the sizing based on calculated percent can be found on the **Style** dialog.

Ignore Dynamic Open Sizing

When checked, any sizing of pie chart/gauges based on the device that is represented being out-of-service will be ignored and the object will stay at the defined **Size**. Options to set the sizing for open devices can be found on the **Style** dialog.

Always Show Value (Percent)

When checked, the options set on the **Style** dialog to display the percent on pie chart/gauges are ignored and the percent value is always shown.

Style, Show Style Dialog

The **Style** defines the set of generic options that can be applied to any pie chart/gauge. An already defined style can be selected from the dropdown. Click **Show Style Dialog** to modify an existing style or to create a new style; this opens up the [Pie Chart / Gauge Style Dialog](#).

Limits

Low Limit, High Limit

Determines the low and high values from which the percent will be calculated and the gauge fill will be drawn. These values are not enterable if the limits are determined from the **Style** or based on device limits. More information on how these limits are used in calculating the percent is given in the **Pie Chart Parameters Tab** and **Gauge Parameters Tab** sections in the **Pie Chart/Gauge Style Dialog** description.

Use Device Low Limit

When checked, the low limit will be determined from the low limit defined for the device for the selected field type. If device limits are not defined for the selected field, this option will not be enabled. If not enabled, the low limit will be based on the value entered in the **Low Limit** box.

Use Device High Limit

When checked, the high limit will be determined from the high limit defined for the device for the selected field type. If device limits are not defined for the selected field, this option will not be enabled. If not enabled, the high limit will be based on the value entered in the **High Limit** box.

Override with Limits from Style

When checked, the high and low limits from the selected **Style** will be used.

Zero Percent Deadband

Use Style Zero Percent Deadband

When checked, the zero percent high limit and zero percent low limits from the selected **Style** will be used.

Zero Percent Low Limit, High Limit

Determines the low limit and high limit for the zero percent deadband from which the percent will be calculated and the gauge fill will be drawn. These values are not enterable if the deadband is determined from the **Style**. The deadband represents the range of values for which the calculated percent will be zero. More information on how the deadband limits are used in calculating the percent is given in the **Pie Chart Parameters Tab** and **Gauge Parameters Tab** sections in the **Pie Chart/Gauge Style Dialog** description.

Pie Chart / Gauge Style Dialog

The **Pie Chart/Gauge Style Dialog** is available from any of the pie chart/gauge dialogs by selecting **Show Style Dialog** or from **Options > Oneline Display Options... > Pie Charts/Gauges** on the **Pie Charts/Gauge Styles** tab.

This dialog lists options that can be commonly applied to pie chart/gauge objects so that these same options do not have to be set for every individual object. The defined styles can be applied to any of the new pie chart/ gauge objects that include Areas, Buses, Generators, Injection Groups, Owners, Substations, Super Areas, Switched Shunts, Transformers, and Zones. These styles are currently not applicable to Lines and Interfaces. Some options are related to options that are set for individual pie chart/gauges. More information about individual object settings can be found in the **Pie Chart/Gauge Dialogs** section.

The [Pie Chart / Gauge Example](#) shows how the fields in this dialog affect the pie chart or gauge display. The items shown in the diagram are explained in more detail throughout this section and in the [Pie Chart / Gauge Dialogs](#) description.

Creating and Saving Styles

Style Name

Drop-down box lists all currently defined pie chart/gauge styles. Select **Add New** to create a new style. Select **Rename** to change the name of an existing style. Select **Delete** to remove an existing style.

Save AXD

Save all currently defined pie chart/gauge styles to a Display Auxiliary file.

Load AXD

Load pie chart/gauge styles from a Display Auxiliary file.

Hide all Style Objects (except in edit mode)

When checked, the pie chart/gauge will not be visible in run mode.

Show Header (Object ID)

When checked, the header of the pie chart/gauge will contain identifying information about the device that the pie chart/gauge represents.

Show Footer (Field Type)

When checked, the field type that is being used in the pie chart/gauge will be displayed in the footer of the pie chart/gauge.

Default Style

When checked, this style is one of the default styles that are defined. Default styles cannot be renamed or deleted. This is an informational field and cannot be changed by the user.

Limits and Zero Percent Deadband Values

Define the High and Low Limits to use when the option **Override with Limits from Style** is selected for the pie chart/gauge that is using this style and the High and Low Zero Percent Deadbands to use when the option **Use Style Zero Percent Deadband** is selected for the pie chart/gauge that is using this style. The deadband represents the range of values for which the calculated percent will be zero. More information is given in the [Pie Chart Parameters Tab](#) and [Gauge Parameters Tab](#) sections on how the limits are used in calculating the percent.

The remainder of the dialog is broken down into four tabs:

- [Standard Parameters](#)
- [Open Parameters](#)
- [Pie Chart Parameters](#)
- [Gauge Parameters](#)

Pie Chart / Gauge Style Dialog - Standard Parameters Tab

Normal Scalar, Normal Fill Color

These are the standard, pre-warning scaling factor and fill color for pie chart/gauges. To change the color, click on the color to open the Color dialog.

Font Color

Font color of the text displayed in the pie chart/gauge. To change the color, click on the color to open the Color dialog.

Border Width

Width of any border portion of the pie chart/gauge. The border of a pie chart encloses the entire pie chart as well as the filled portion of the pie chart. The border of a gauge encloses the entire gauge. The border width also impacts the width of the lines marking the limits and deadband on the gauge.

Border Color

Color of any border portion of the pie chart/gauge. The border of a pie chart encloses the entire pie chart as well as the filled portion of the pie chart. The border of a gauge encloses the entire gauge. The border color also is the color used for the lines marking the limits and deadband on the gauge. To change the color, click on the color to open the Color dialog.

Border Color Same as Fill Color

When checked, the border color will be the same color as the fill color and the color selected in the Border Color will be ignored.

Background Color

The drop-down box allows selection of three options for the background color: Oneline Background, Specific Color, and Clear. Oneline Background will color the pie chart/gauge background the same color as the selected background color for the oneline. Specific Color will allow the selection of a user specified color. To specify a color, click the box to the right of the drop-down to open the Color dialog. The Clear option will not use a background color and objects underneath the pie/chart gauge will be visible through the unfilled portions of the pie chart/gauge.

Relative Font Size

Modify the track bar to adjust the size of the font relative to the size of the pie chart/gauge.

Max Zoom Percent for Full Resize

Pie chart/gauges dynamically resize when zooming in and out on oneline diagrams. A limit can be placed on the point at which the pie chart/gauges resize when zooming in by setting a zoom level in this field. This helps prevent the pie chart/gauges from getting so large that they occupy the entire screen.

Warning/Limit Scalars and Colors

The table in this section allows the definition of different settings for pie chart/gauge size and fill color based on the percent value represented by the pie chart/gauge. Points can be added or removed from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color dialog.

Use Discrete Map

When checked, discrete colors will be used for the pie chart/gauges based on the colors specified in the table. Otherwise, colors will be blended if the percentage represented in the pie chart/gauge falls between percentages defined in the table.

Pie Chart / Gauge Style Dialog - Open Parameters Tab

Open parameters are applied to any pie chart/gauges when the device that they represent is out-of-service.

Special Formatting for Open Devices

When checked, the **Open Size Scalar**, **Open Border Width**, and **Open Border Color** options are applied when a device is out-of-service.

Use Special Open Background Color

When checked, the **Open Background Color** option is applied.

Open Symbol

When an **Open Symbol** other than None is selected, the symbol will be drawn in the selected **Open Symbol Color**.

Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab

Options given on this tab only apply to how pie charts are displayed.

Show Value Percent

When the percent of the pie chart/gauge exceeds the value specified in this field, the percent is shown as text within the pie chart/gauge. The default is 80%.

Display of Values Below the Zero Percent Deadband

The selection of this option determines how to calculate the percent represented in pie charts. Four parameters are required for determining this: low limit, high limit, zero percent low limit, and zero percent high limit. These four limits are determined from options set with individual pie charts and/or options set with the style. In general, the zero percent low limit and zero percent high limit define the deadband in which the percent is zero. The percent is then determined based on where the selected field value falls within the high limit and zero percent high limit and the low limit and zero percent low limit. The specific calculations used for determining the percent is given for the options below. In all cases if the limits are not defined correctly (i.e., none of the If..then statements below apply), the percent will be returned as zero.

- **Allow negative percent values**
 - oIf (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then
Percent = ((field value) – (zero percent high limit))/((high limit) – (zero percent high limit))
 - oIf (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then
Percent = ((field value) – (zero percent low limit))/((zero percent low limit) – (low limit))
- **Use absolute value** (i.e., percent always >= 0)
 - oSame calculations as in **Allow negative percent values** except that the absolute value of the calculated percentage is reported.
- **Low limit is zero percent**
 - oThe zero percent deadband limits are ignored in this option and the percent is calculated based on the low and high limits only
 - oIf (high limit) > (low limit) then
Percent = ((field value) – (low limit))/((high limit) – (low limit))
- **Treat values below zero deadband as zero**
 - oIf (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then
Percent = ((field value) – (zero percent high limit))/((high limit) – (zero percent high limit))
 - oIf (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then
Percent = 0

Pie Chart / Gauge Style Dialog - Gauge Parameters Tab

Options given on this tab only apply to how gauges are displayed.

Show Limit Labels

When checked, text labels will be drawn with the gauge showing the four limits in use.

Include Units in Limit Labels

When checked, units will be included when the limit labels are shown.

Numeric Value Type

This option allows the numeric value to either be displayed as the **Actual Value** or the calculated **Percent** value.

Display of Numeric Value

This option determines when the value of the field is displayed with the gauge. **Never** will not show the value, **Always** will always show the value, and **If Violating Warning/Limit Value** will only show the value if it is outside of the limit range that is currently in use.

Digits to Right of Decimal

This parameter determines how many digits to the right of the decimal are displayed in the limit labels and the numeric value when they are shown with the gauge.

Gauge Width is Always a Fixed Proportion of Size

When checked, the **Gauge Fixed Width/Size Ratio** is used to determine the width of the gauge as a proportion of the size (height). Note: the size of gauges is specified for individual gauges on the pie chart/gauge dialogs.

Gauge Fixed Width/Size Ratio

Specifies the ratio of the gauge width to the size when the **Gauge Width is Always a Fixed Proportion of Size** option is checked.

Percent Overhang on Top of Gauge

The overhang at the top of the gauge is the region above the high limit line. The size of this region is defined as a percentage of the total size of the gauge.

Percent Overhang on Bottom of Gauge

The overhang at the bottom of the gauge is the region below the low limit line. The size of this region is defined as a percentage of the total size of the gauge.

Gauge Fill and Percent Calculation

These options determine how the calculated percent will be displayed and how the gauge will be filled. Four parameters are required for determining these: low limit, high limit, zero percent low limit, and zero percent high limit. These four limits are determined from options set with individual gauges and/or options set with the style. In all cases if the limits are not defined correctly, the percent will be returned as zero.

Use Absolute Value for Percent

When checked, the displayed percent will be the absolute value of the calculated percent. This option does not impact how the gauge will be filled.

Gauge Fill Options (Zero Percent Reference)

This option determines how the gauge will be filled. If **Fill up or down from deadband** is selected, the fill will begin at the low or high deadband limit and continue up or down depending

upon where the field value falls within the defined limits. If **Fill up from bottom** is selected, the fill will begin at the bottom of the gauge and fill up to the field value.

Regardless of which fill option is selected, the following equations will be used to calculate the percent based upon where the field value falls within the defined limits:

If (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then
 Percent = ((field value) – (zero percent high limit))/((high limit) – (zero percent high limit))

If (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then
 Percent = ((field value) – (zero percent low limit))/((zero percent low limit) – (low limit))

Ignore Deadband

When checked, deadband limits are ignored. This effectively forces the fill to start from the bottom of the gauge and fill up to the field value. The percent is calculated based on the following equation and will always be reported as the absolute value:

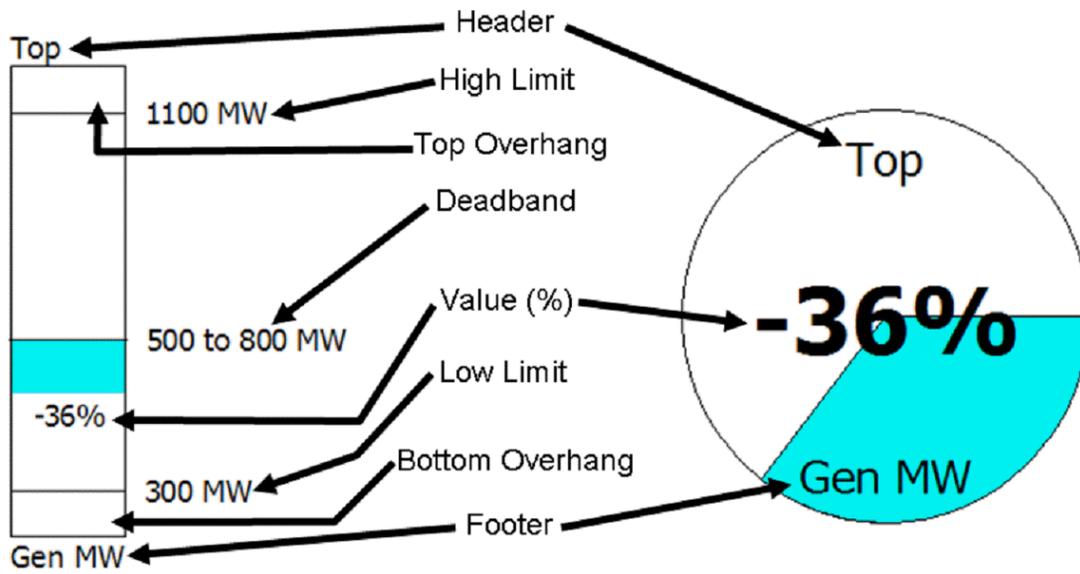
If ((high limit) – (low limit)) <> 0 then
 Percent = ((field value) – (low limit))/((high limit) – (low limit))

Override Low Limits with a Value of Zero

When checked, the low limit value will be set to zero when determining the fill and calculating the percent. This option is only applied if the **Ignore Deadband** check box is also checked. The percent is calculated based on the following equation and will always be reported as the absolute value:

If ((field value) > 0) and ((high limit) <> 0) then
 Percent = (field value)/(high limit)

Pie Chart / Gauge Example



Chapter 5: Edit Mode Tools and Options

Edit Mode provides a number of commands for customizing and modifying the case that are different from Run Mode.

This chapter covers the following:

- General Edit Mode Tools
- Insertion Using Palettes
- Automatic Object Insertion
- Equivalencing
- System Scaling
- Facility Analysis

Appending a Case

The Append Case command allows you to append additional power system components to an existing case. Unlike the [Open Case](#) command, Append Case does not delete the existing case (if any) before loading the selected case. To append a case to the existing case, go to the [Tools](#) ribbon tab and choose **Modify Case > Append Case** from the [Edit Mode](#) ribbon group.

The Append Case command can be useful when used in conjunction with the [Equivalencing Display](#).

Notes

Appended data completely overwrites an existing record corresponding to the same bus number(s). Branch elements are appended to a case only if both of their terminal buses exist.

Refresh Anchors

The **Refresh Anchors** option of the [Formatting](#) ribbon group on the [Draw](#) ribbon tab allows you to update or reset the anchoring of objects on a oneline diagram to their respective anchor. It can be advantageous to use the Refresh Anchors option when opening onelines that were created by another case, copying and pasting data between onelines, or renumbering objects on the oneline. This ensures the anchoring of objects to the appropriate anchor on the oneline diagram.

Bus Renumbering Dialog

The Renumber Buses Display allows you to change the bus numbers for either the entire power system case, and/or for any open oneline diagrams. To show this display, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group.

The bus renumbering feature is provided because it is sometimes necessary to renumber buses in the power system model, either to make room for new buses or to move buses to a different zone or area that has a different numbering scheme. It may also be necessary to renumber bus display objects on the oneline if you want to use the oneline with a case other than the one for which it was originally designed. The bus renumbering feature provides a convenient way of accomplishing this.

The table at the bottom of the Bus Renumbering Dialog is used to manage the lists of current bus numbers and any desired changes to the numbering scheme. You can specify the bus numbers to change and their new values by directly typing them into the table. Alternatively, you can generate the bus list automatically by selecting one of the [Automatic Setup of Bus List Options](#) and clicking the **Setup Bus Swap List** button. The Automatic Setup options allow you to add to the list all buses in the case, all buses in the case subject to the [area/zone/owner filter](#) constraints, all buses currently displayed on the oneline, or a set of numbers from a text file.

The table behaves just like a [Case Information Display](#) and thus has a [local menu](#) that can be invoked by clicking the right mouse button. Select **Insert** from the local menu to insert a new bus number to change, or select **Delete** to delete a bus number that is currently in the list. Select **Show Dialog** to display the [Bus Information Dialog](#) corresponding to the bus number on which you right-clicked. You may clear the entire list by pressing the **Clear Bus List** button.

Once you have indicated which buses you would like to renumber in the table, select an option from **Bus Change Options** to specify where you would like to implement the changes (in both the case and the oneline, in the case only, or in the oneline only). Make the changes by pressing the **Change Bus Numbers** button. Close the dialog by pressing **Close**.

The Automatic Setup of Bus List and Bus Change options deserve further discussion. See for further details or click [NEXT](#).

Bus Renumbering: Automatic Setup of Bus List Options

This section of the [Bus Renumbering Dialog](#) is used to generate automatically a list of the buses to be renumbered. Select an option from the list and click on the **Setup Bus Swap List** button to generate the list, or **Clear Bus Swap List** to clear the list.

Load All Buses in Cases

Creates entries in the table for every bus in the system. By default the new bus number is the same as the old bus number. Of course you do not have to renumber every bus. If you would like a bus to keep its same number, either add that bus to the table, or simply have an entry with the old and new bus numbers identical.

Load Buses with Valid Area/Zone Filters

Same as the **Load All Buses In Case** option except only those buses with valid [area/zone/owner filters](#) are added. This allows you to easily modify the bus numbers in just an area or zone.

Load Only Buses on Oneline

Creates entries in the table only for those buses on the current oneline. This option is most useful when you are just changing the buses on the onelines.

Load Buses From Text File

Creates entries in the table using an external text file. The format of this file is as follows:

```
1 11 Specifies converting from old bus number 1 to new bus number 11
2 22 Specifies converting from old bus number 2 to new bus number 22
etc.
```

Freshen Current Oneline

The Freshen Current Oneline option is designed to help you quickly renumber an existing oneline to work with a new numbering scheme. You will find this method helpful if you have been using the oneline with a case and now must use it with a different case having a different set of bus numbers, but the same bus names. Freshen Current Oneline will try to match the buses on the oneline with the buses in the new case by matching bus names and kV, rather than by number (which is how Simulator usually tries to link bus display objects with bus records in the case). The best way to learn how to use Freshen Current Oneline is to consider the following example:

To update an old oneline to work with a new bus numbering scheme:

- Open the oneline and the old case with which it was used.
- Choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab.
- Select **Load Only Buses on Oneline** and press the **Setup Bus Swap List** button.
- Change the **Swap?** field values all to *Yes*.
- Right click on the table and choose **Save List to File**. Give the file a name. For this example, we'll name the file "oldscheme.txt." This file will contain the list of buses represented on the oneline, specifying each bus's number, name, kV, and area.
- Close the old case.
- Open the new case and the oneline you wish to renumber. If any other onelines open with the case, close them. You want only the oneline you wish to renumber to be shown.

- Choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab.
- Select **Freshen Current Oneline** and specify the file "oldscheme.txt" (for this example). Click the **Setup Bus Swap List** button. Simulator will match the old numbering scheme used in the oneline with elements in the new case by name and kV. If it finds more than one match, it will use the element's area name as a tie breaker. If it still can't reconcile the multiple matches, it will add both renumbering options to the table.
- Go through the new list and make sure that you want to swap the buses that are listed. If you do, change the **Swap?** field value for each to *Yes* (you can do this quickly for all buses by right-clicking on the **Swap?** column and choosing **Toggle All Yes**). Be sure to reconcile any duplicate bus renumbering suggestions. These are cases for which Simulator could not determine how to renumber the buses because a bus on the diagram matches more than one bus in the case by name, kV, and area.
- Click the **Change Bus Numbers** button at the bottom of the form.

Once the oneline has been renumbered, save it with the case by selecting **Save Case** (or **Save Case As** if you wish to give it a different name) from the [Application Button](#). See [Bus Change Options](#) for additional details.

Bus Renumbering: Bus Change Options

This option is used to specify which buses to change.

Change Both Case and Onelines (default)

Renumbers the buses in both the case and any open oneline diagrams.

Change Only Case

Only renumbers the buses in the case. The oneline bus numbers are not changed.

Change Only Onelines

Only renumbers the buses on the onelines. The case itself is not changed. You would want to select this option if you have already changed the case (or loaded a different one), but now have several onelines based on that case that also need to be changed. See [Automatic Setup of Bus List Options](#) for more details. This is the most commonly used option.

Renumber Areas/Zones/Substations Dialog

The Renumber Areas/Zones/Substations Dialog is available while in [Edit Mode](#), by going to the [Tools](#) ribbon tab, and choosing **Renumber > Renumber Areas/Zones/Substations** from the [Edit Mode](#) ribbon group. This dialog allows the renumbering of areas, zones, and substations for either the entire power system, and/or for any open oneline diagrams. This dialog is similar to the [Renumber Buses Dialog](#).

This feature is provided because it is sometimes necessary to renumber areas, zones, or substations in the power system model, either to make room for new elements or to move elements to different area, zones, or substations that have a different numbering scheme. It may also be necessary to renumber display objects on the oneline if the oneline is used with a case other than the one for which it was originally designed. The areas/zones/substations renumbering feature provides a convenient way of accomplishing this.

The table at the bottom of the Renumber Areas/Zones/Substations Dialog is used to manage the list of areas, zones, and substations available for renumbering. This list is populated by selecting one of the **Automatic Setup of Swap List** options. The Automatic Setup options allow the addition of all areas, zones, or substations in the case or only those on the oneline. An option is also available to read the swap information from a file or to setup the swap list based on the current oneline. The **Types to Insert** option is available for specifying whether or not to add areas, zones, and substations to the swap list. Once the swap list option and types to insert have been selected, click the **Setup Swap List** to populate the table.

The table behaves just like a [Case Information Display](#) and thus has a [local menu](#) that can be invoked by clicking the right mouse button. Select one of the **Insert** options from the local menu to insert a new area, zone, or substation number to change, or select **Delete** to delete an element number that is currently in the list. The entire list may be cleared by clicking the **Clear Swap List** button. The **Present Number** for the element is specified in the list. Change the **New Number** field to specify the updated number of the area, zone, or substation.

The **Renumber Options** allows the specification of where the renumbering changes should be implemented. The changes can be applied to the power system case only, the oneline only, or both the case and the oneline.

Once the elements to renumber have been specified and the **New Number** fields have been set appropriately, implement the changes by pressing the **Renumber** button. The changes will be applied to the case and/or oneline as specified in the **Renumber Options**. Close the dialog by pressing **Close**.

Merging Buses

Two or more buses can be merged to a new bus in Edit Mode with the loads, generators, and shunts of the buses merged moved to the new bus. The transmission lines among the buses merged will be deleted while the transmission lines connecting the merged buses and buses which are not selected to merge will be moved to the new bus.

To join two or more buses together using the oneline display, select at least two power system elements (at least one of them being a bus), then right click on one of the selected elements to invoke the popup menu and select **Merge selected buses**. In the Bus Merging Dialog, specify the buses to merge further if needed by clicking **Add** and/or **Delete** button. To add buses to merge, click **Add** and select all the needed buses in the **Choose a Bus** dialog and click **OK**. To delete from already selected buses, select the buses to delete and click the **Delete** button.

Once you have the elements to merge selected, enter the number, name, nominal voltage, zone number, area number and substation number for the new bus. These properties can be set to be the same as one of buses to merge by selecting it in the **Specify buses to merge** box and pressing the **Set new bus properties same as selected bus**.

Buses can also be merged in the [Bus Case Information Display](#). To do so, select a cell and right click to popup the local menu. Select **Merge selected buses**. The buses to merge and the properties of the new bus can be edited in the Bus Merging Dialog.

When buses are merged from the oneline diagram, the selected buses will be joined in both the PWB case and the oneline. When buses are merged from the bus grid, the selected buses will be joined only in the PWB case but not in the oneline. This might result in bus objects not connected to bus records in the oneline.

Splitting Buses

Simulator assists you in transforming one bus into two connected buses. This process is called splitting a bus. In performing the split, the user is able to decide which equipment to keep connected to the original bus and which equipment to move to the new bus. Because this activity impacts the structure of the power flow equations, bus splitting can be performed only in [Edit Mode](#).

To split a bus from the oneline diagram, right click on the bus you wish to split, and click **Split Bus** from the resulting local menu. To split a bus from the [Bus Case Information Display](#), right-click on its corresponding record and again click **Split Bus** from the resulting local menu. In either case, the [Split Bus Dialog](#) will appear.

[Multi-section lines](#) merit special consideration during bus split operations. These are the rules Simulator follows when you try to split a bus that is part of a multi-section line. If the original bus is the endpoint bus of a multi-section line and the ending line segment was transferred to the new bus, then the new bus becomes the new ending terminal of the multi-section line. If the original bus was a dummy bus of the multi-section line, and if exactly one of the branches connected to the original bus is rerouted to the new bus, then both the new bus and the original bus will be dummy buses in the reconstituted multi-section line. If neither or both of the lines connected to the original dummy bus were rerouted to the new bus, then the multi-section line definition is eliminated, since Simulator has no way to determine how the multi-section line should be redefined.

A final consideration involving bus splits is how [sequence data](#) is treated. If you have defined sequence data for fault analysis, Simulator will recalculate the sequence data for the original and new buses after the split. In this case, the zero sequence impedance for the new branch that connects the original and the new bus will be set to $j0.0001$.

Split Bus Dialog

The Split Bus dialog contains two sections labeled **Existing Bus** and **New Bus**. These sections enable you to designate the bus you want to split and the name and number of the new bus created by the bus split operation. If you opened the Split Bus Dialog from either the [Bus Case Information Display](#) or an oneline diagram, the Existing Bus Name and Number fields will be read-only and will identify the bus you selected. However, if you opened the Split Bus Dialog from the **Modify Case Menu** on the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab, you will first have to choose a bus to split. In this case, use the **Find...** speed button located next to the **Existing Bus** label to select the bus to split, or simply type its number in the Existing Bus Number field. Then, provide the name and number for the new bus to create using the New Bus Name and Name fields. The number you specify in the New Bus Number field must be unique; it cannot be a number that identifies another bus in the case. If you do specify an existing number, Simulator will issue an error message and require you to specify a different number. If you choose not to specify a name for the new bus, Simulator will set the name of the new bus to be the same as the bus's number.

Once you have identified the bus to split and the name and number for the new bus resulting from the split, you may then specify whether a bus tie should be inserted between the existing bus and the soon-to-be-created new bus. By checking the **Insert bus tie between existing and new buses** checkbox, you command Simulator to place a very low-impedance bus tie between the bus to split and its offspring. The new branch will have an impedance of $0.0000 + j0.0001$ ohms. If the bus tie should be inserted as an open branch, check the **Normally open** checkbox. This option becomes available, of course, only if you elect to have Simulator automatically add the new bus tie.

After you have finished making your selections, click **OK**. Simulator will create the new bus, assign its electrical attributes to match those of the existing bus, and add the bus to the power system model. If a oneline diagram is currently active and the existing bus is represented on it, Simulator will add a symbol for the new bus to the diagram, placing it immediately to the right of the existing bus's symbol. If you elected to create a bus tie between the existing and new buses, Simulator will also add a symbol for the bus tie to the diagram.

Finally, Simulator will automatically open the [Equipment Mover Dialog](#) to help you manage the transfer of equipment from the existing bus to the new bus.

Equipment Mover

Simulator provides a convenient tool for transferring equipment between buses. Simulator allows you to move bus shunts, loads, generators, switched shunts, and transmission lines between buses. For loads, generators, and both varieties of shunts, Simulator offers you the ability to transfer all or part of the equipment from the origin bus to the destination bus.

Equipment may be transferred between buses using the Equipment Mover Dialog. The Equipment Mover Dialog can be opened in any of four ways:

- Go to the [Tools](#) ribbon tab and choose **Modify Case < Move Bus Equipment...** from the [Edit Mode](#) ribbon group.
- On the oneline display right-click on the bus from which you want to move equipment and select **Move Equipment...** from the resulting local menu.
- From a [Bus Case Information Display](#), right-click on the record corresponding to the bus from which you want to move equipment and select **Move Equipment ...** from the resulting local menu.
- As the final step of the [Split Bus](#) operation.

Regardless of the approach you take, Simulator will then open the Equipment Mover Dialog. The Equipment Mover Dialog consists of three sections. The top portion of the dialog is split in two sections that identify the bus from which equipment will be transferred (on the left) and the bus equipment will be transferred to (on the right). If the dialog was opened as the final operation of the [Split Bus](#) operation, these two buses will be hard-coded to identify the original bus and the bus resulting from the split. If the dialog was opened from the main menu, you must select both the origin and destination buses from lists that are reminiscent of [case information displays](#). If the dialog was opened using one of the other two methods, the origin bus will be set to the bus you selected, and you will then have to choose the destination bus. To select a bus to be an origin or destination bus, simply select the corresponding record from the appropriate list.

Once the origin and destination buses have been identified, you must then select the equipment to transfer from the origin to the destination bus using the case information display that occupies the bottom of the dialog. To move a particular piece of equipment, toggle the value of the **Move Object?** field to *YES*. To move just portions of loads, generators, or shunts from the origin to the destination bus, adjust the value of the **Move %** field from 100.0 to the percentage you desire.

Once you have selected the equipment you wish to transfer, click the **Move equipment** button. Simulator will adjust the power system model to reflect your equipment transfer requests. Furthermore, Simulator will provide you an opportunity to manually adjust all open oneline diagrams to reflect the equipment transfers. To facilitate this activity, Simulator will open a [Potential Misplacements Dialog](#) for each oneline that displays the origin bus. The Potential Misplacements Dialog lists the display objects associated with the equipment that had just been transferred from the origin to the destination bus. By clicking on an entry in this list, you can pan the associated oneline diagram to focus on that object. This allows you to identify display objects that perhaps should be relocated to reflect their new bus associations. Once you have finished addressing these potentially misplaced display objects, click the **OK** button to close the Potential Misplacements Dialog.

If you find that you would like to reopen the [Potential Misplacements Dialog](#) after you have closed it, click the **List most recent transfers** button. This will reopen the Potential Misplacements Dialog associated with the most recent equipment transfer operation.

To conclude the equipment transfer operation, click the **Close** button.

Potential Misplacements Dialog

In the wake of an equipment transfer operation using the [Equipment Mover Dialog](#), some oneline display objects may be out of place. To ease the task of correcting these misplacements, Simulator provides the Potential Misplacements Dialog. Simulator opens a Potential Misplacements Dialog for each oneline that displays the bus that served as the origin for equipment transfer. The Potential Misplacements Dialog lists the display objects associated with the equipment that had just been transferred from the origin to the destination bus. By clicking on an entry in this list, you can pan the associated oneline diagram to focus on that object. This allows you to identify display objects that perhaps should be relocated to reflect their new bus associations. Once you have finished addressing these potentially misplaced display objects, click the **OK** button to close the Potential Misplacements Dialog.

Tapping Transmission Lines

Simulator eases the process of inserting a bus at some location along an existing transmission line. This feature can be extremely useful when you want to add a new generation site to a model, for example. Rather than having to delete an existing line, place the bus, and draw two new transmission lines, Simulator simplifies the task to a one-step process.

A line can be tapped from a oneline diagram, the transmission line case information display, or by going to the [Tools](#) ribbon tab and choosing **Modify Case > Tap Transmission Line** from the [Edit Mode](#) ribbon group. From the oneline diagram, right-click on a transmission line and select **Insert Line Tap** from the popup menu. From the [case information display](#), simply right-click on the corresponding branch record and select **Tap Transmission Line** from the popup menu. Any of these methods will open the [Automatic Line Tap Dialog](#) for setting up and inserting the new bus.

Note that transmission lines may be split only from [Edit Mode](#). You cannot access this functionality from [Run Mode](#).

Automatic Line Tap Dialog

The Automatic Line Tap Dialog will allow you to define the settings to use for inserting a new bus along a transmission line. The transmission line to tap will be represented in the panel at the top of the dialog, displayed as a **Near Bus** and **Far Bus**. If the Automatic Line Tap Dialog was opened by right-clicking a line on a oneline diagram, or by right-clicking on a record in the case information display, the line will already be selected in the panel. You can change the line selection by first choosing the **Near Bus** you desire, and then selecting the **Far Bus** from the list of possible connections to the chosen **Near Bus**. Note that the percentage entered in the **Position along line** field will be in relation to the selected **Near Bus**. The inserted bus and new sections of the tapped line will adhere to the following settings:

Position along line

The field labeled **Position along line** will indicate the point where you right-clicked the mouse relative to the location of the Near Bus to the mouse-click in terms of a percentage of the total line length. In other words, if the line is 10 units long, and you clicked the right mouse at location 7 units from the Near Bus end of the line, the **Position along line** will indicate 70%. If you opened the Automatic Line Tap Dialog from the case information display or the **Tools** ribbon tab, the **Position along Line** will be set to 50%. In either case, you can adjust this setting to place the new bus more precisely. The placement of the new bus controls how the impedances of the new lines are set, as the impedance of each section will equal the section's corresponding percentage length multiplied by the impedance of the original line. Note that the original charging capacitance of the line will be reassigned as determined by the selection under the **Shunt Model** option.

New Bus Number

By default, Simulator will find and set an unused bus number for you, but you can specify the number to be used for the new bus, between 1 and 2147483647. If you enter a bus number that already exists, you will be prompted to enter a different number when you click the **Tap** button.

New Bus Name

Specify a name to be assigned to the new bus. By default this field is blank, and if left blank Simulator will set the name of the bus the same as the new bus number.

New Bus Area

You can specify the area for the new bus to be the same as the **Near Bus**, the **Far Bus**, or another value of your specification. If you select the **Specify** option, the edit box and find button will become enabled. You can then enter an area number manually, or click **Find** to locate an area from the list of areas currently in the case. If you want a new area to be assigned to the case for this bus, simply enter an unused area number manually in the box, and Simulator will automatically set up the new area record for the case. You can then open the [Area Information Display](#) and set the name and other values for the new area.

New Bus Zone

You can specify the zone for the new bus to be the same as the **Near Bus**, the **Far Bus**, or another zone of your specification. If you select the **Specify** option, the edit box and find button will become enabled. You can then enter a zone number manually, or click **Find** to locate a zone from the list of zones currently in the case. If you want a new zone to be assigned to the case for this bus, simply enter an unused zone number manually in the box, and Simulator will automatically set up the new zone record for the case. You can then open the [Zone Information Display](#) and set the name and other values for the new Zone.

Shunt Model

By default, Simulator will reassign the original charging capacitance as line shunts at the original terminal bus ends of the two new line segments. The charging capacitance of the two new branch elements will be set to 0. This will result in an exact match of power flows before and after the line tap.

Alternatively, you can choose to have the capacitance converted approximately using percentage entered as the **Position along line**. In other words, the original capacitance will be scaled and assigned to the two new segments as charging capacitance. This will result in a slightly different power flow result after the line tap, as it is an approximation of charging capacitance on the two new segments.

Treat sections as a multi-section line

Check the box labeled **Treat sections as a multi-section line** to force the status of the two new line sections to be controlled in unison. Checking this box will cause a multi-section line to be created comprised of the two new line sections. See [Multi-Section Line Information](#) for details.

Click the **Tap** button to close the Automatic Line Tap Dialog and perform the line tap. If a value on the dialog is not set properly, a warning message will appear, and you will need to either change the specified value or cancel the process.

Setting Default Drawing Options

The Default Drawing Options Dialog is used to define the default sizes of new display objects, as well as various other display parameters. These options are available only in Edit Mode.

To open this dialog choose **Default Drawing Values** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Online](#) ribbon tab.

The Default Drawing Values Dialog is organized using a list showing the types of objects that can be added to onelines. To view information about a particular type of object, select that object type from the list and tables which describe the defaults available for that object will be populated. Two tabs will be available. One will contain the majority of default options that are available for the selected display object type, and the other will contain the default positions of fields that will be inserted when the object is inserted.

Generally, the Default Drawing Values apply when you insert *new* objects on an oneline diagram. Changing the Default Drawing Values does not automatically affect the *existing* objects already drawn on the diagram. However, you can select multiple objects on a oneline diagram, and right-click on any one object to invoke the object's popup menu. You will find the option **Apply Default Draw Values**, which will modify the formatting of the selected objects to meet the default drawing value specifications.

Variable Defaults depending on Voltage Level

Many kinds of objects can have more than one set of defaults specified depending on their voltage level. When more than one set of defaults is specified, the table is always sorted by Nom kV.

A new object is inserted with properties corresponding to the default with the lowest Nom kV that the object's nominal kV is larger than. If the object's nominal kV is smaller than all defaults, then it will be set according to the default with the lowest nominal kV.

For instance, assume 4 sets of defaults are defined for buses roughly as follows

Nom kV Size

>400 15

>300 10

>200 8

>100 5

A new bus with nominal voltage of 345 kV is inserted with size 10.

A new bus with nominal voltage of 299 kV is inserted with size 8.

A new bus with nominal voltage of 69 kV is inserted with size 5.

In order to add another set of defaults for a kind of object, right-click on the table and choose **Insert** from the local menu. In order to delete a set of defaults, right-click on the table and choose **Delete**. Once you have inserted a new default, specify the new Nom kV for the default along with the new defaults.

Notes: When you change the Nom kV of a default, the list always sorts itself according to Nom kV. **Insert** will not be enabled in the local menu for objects that cannot have more than one set of defaults. **Delete** will not be enabled if only one set of defaults is defined.

Modifying Values in the Default Drawing Values Dialog

The font colors in tables of this dialog follow the conventions of the [Case Information Displays](#). Most default values are enterable and can be modified by selecting the value you want to change and

then typing in the new value. Other values are toggleable and can be changed by double-clicking on the value. For reference regarding what the various "size" defaults mean, when the zoom level is at its nominal value (100%), the size of the screen is 100 by 100. There are also some special kinds of fields in this dialog which are discussed next.

Editing Colors and KV-Based Color Coding

In order to modify a color value, double-click on the colored rectangle to bring up the Color Dialog. Then specify the color you want. For some defaults, such as fill color, it is possible to specify the default color to be *none*. In order to set a color default to *none*, select the colored rectangle and press the Delete key.

At times, a user may want to see the kV level of screen objects directly from the oneline diagram, without having to open a text display. One way to do this is to color code the elements on the oneline diagram according to kV level. Right-click on the table and choose **Insert** from the local menu. Specify the Nom kV and color.

Editing Default Field Positions ("Pos1", "Pos2", etc...)

Many objects allow you to specify some default fields to be automatically added when a new object is drawn. For example, you may want to insert the bus name next to all new Bus Objects. For objects that allow the insertion of fields, a tab with field position diagrams will be available when that object type is selected. In order to specify the default positions of fields, click on the tab showing the field positions and click on the table row corresponding to the Nom kV of the defaults of interest. For object types that do not allow specifying the defaults by Nom kV, simply update the given position diagram. The position diagram will show the present settings for the new fields. Positions that have a default specified will be highlighted and the name of the default field will be shown. Positions with no default field will not be highlighted and will say "Pos1", "Pos2", etc.... For objects that have more than one possible orientation (e.g. generators can be up, right, left or down), there are multiple position diagrams showing the positions for each orientation.

In order to modify the settings, move your mouse over the position on the dialog you want to change and click. This will bring up the appropriate Field Options Dialog such as the [Bus Field Options](#), [Generator Field Options](#), [Load Field Options](#), [Switched Shunt Field Options](#), [Line Field Options](#), [Interface Field Options](#), [Substation Field Options](#), [Area Field Options](#), [Super Area Field Options](#), or the [Zone Field Options](#). Simply select the field you want and choose **OK**. If you would like to set a default field to "none", click **Remove Field** instead of **OK**. You can also modify default fields by double-clicking on the table for "Pos1", "Pos2", etc... This brings up the appropriate Field Options Dialog as well. If you press the Delete key while on a "Pos" field in the table, it will set the default field to "none".

Editing Stub Size and Stub Space

When utilizing the [Automatically Insert Transmission Lines](#) feature, the **Stub Size** and **Stub Space** values are used. Simulator will draw each automatically inserted branch such that it emerges from both its terminal buses at right angles. To accomplish this, Simulator draws each automatically inserted branch in three segments: two stubs perpendicular to the terminal buses having a length specified by the value supplied for **Stub Size**, and a third segment joining the two stubs. The amount of space between lines as they converge into a bus is set by **Stub Space**.

If you do not want transmission line stubs to be inserted, then set **Stub Size** to *none* or to a negative number. Note in order to set a value for **Stub Space**, **Stub Size** must be set to a positive number first.

Editing CB Size

When you insert new transmission lines or transformers, circuit breakers will automatically be inserted on the branch with their size specified by CB Size. If you do not want Circuit Breakers inserted for a particular voltage level, then set CB Size to *none* or a negative number.

Editing Pie Size

When you insert new transmission lines or transformers, pie charts will automatically be inserted on the branch with their size specified by Pie Size. If you do not want pie charts inserted for a particular voltage level, then set Pie Size to 'none' or a negative number.

Set Default Font

Click on this button to set the default font used for new text fields. This is the font which will be used for all new text fields. Note however that the default font size set using this dialog is not used *unless* the option **Use the default font size for new text objects** is selected. Otherwise, the specific kind of object and voltage level specifies what the font size should be. For example, in the Interface tab, one of the defaults is Font Size. New Interface Fields will use this font size.

Only Cut/Copy Display Objects, Not Power System Records

When you cut or delete an object from the oneline, Simulator needs to know whether you simply want to delete the display object from the oneline or to purge the definition of the power system object from the model. Check this box to tell Simulator to assume that it should always delete objects just from the oneline display, not from the power system model.

Online / Bus View Background Color

To change the default oneline background color, click on the Change button to select a new color. This color applies to both new oneline diagrams and the background color of the [bus view oneline](#) diagram.

Recommended Multi-KV Level Defaults

Click this button to change the default options back to Simulator recommended values. Multiple sets of default options will be specified based on voltage level for objects for which it is beneficial to do this (i.e. transmission lines).

Recommended Single-KV Level Defaults

Click this button to change the default options back to Simulator recommended values. A single set of default options will be specified for each object type.

Bus Selection Page

The Bus Selection page is currently used with two displays:

- The [Equivalencing Display](#), to partition the system into the study subsystem ("Study") and the external subsystem ("External").
- The [Facility Analysis Dialog](#), to determine the buses in the external subsystem from which the buses in the facility would be isolated.

The following description concerns the Equivalencing Display, but the functionality of the Bus Selection Page is similar for both applications.

Buses, Areas, and Zones lists

For both equivalencing and facility analysis, use these lists to manually change the system designation of individual buses, areas, or zones. Note that changing the system designation on the area or zone tab is just another form of changing the designation of study / external system of individual buses. Click on the **Which System?** field in each of these tables to toggle the object's affiliation with the study or external systems, subject to the values of the **Filter by kV** option and the **Include how many tiers of neighbors?** field.

In addition, the bus selection table for [facility analysis](#) contains one additional column labeled **Selected**. This Yes or No field indicates which buses in the Study system belong to the Facility being analyzed. Double-clicking one of the fields in this column will toggle the value of the field between *Yes* and *No*.

External

Use these fields to specify a [range of areas, zones, or buses](#) to be added to the external system. The **Filter by kV** and **Include how many tiers of neighbors?** controls will also shape the selection of buses to add to the external system.

Study

Use these fields to specify a [range of areas, zones, or buses](#) to be added to the study system. The **Filter by kV** and **Include how many tiers of neighbors?** controls will also shape the selection of buses to add to the study system.

Include how many tiers of neighbors?

This value indicates the number of tiers of neighbors to carry with each selected bus when adding the selected bus to either the study or external system. For example, if Neighbor Tiers is 1 and we elect to add bus X to the external system, both bus X and its first tier of neighbors will be added to the external system. If Neighbor Tiers is 0, only bus X will be added to the external system.

Filter by kV

If the **Filter by kV** box is checked, then only buses having a nominal voltage level between the values given in the **Max kV** and **Min kV** fields can be selected.

Set All As External

Click this button to assign all buses to the external system.

Set Branch Terminals External or Study

Clicking either of these two buttons allows setting advanced filter criteria which define a branch or group of branches. Once the criteria are set, clicking **Filter** will select the terminal buses of all branches meeting the filter criteria, and set those buses to either the External or Study system, depending on which button was pressed.

Select Buses using a Network Cut

A custom [network cut](#) can be defined for choosing which buses should remain in either the Study or External system.

Save Buses to File

Once the system has been partitioned, this command allows you to store the numbers of the buses of the study system in a text file.

Load Buses from File

Click on this button to load a listing of the buses to be included in the study system from a text file. You will be prompted to select the text file. The format of this text file is such that one bus number occupies each line. Any buses not identified in this file are defined as being in the external system.

Example

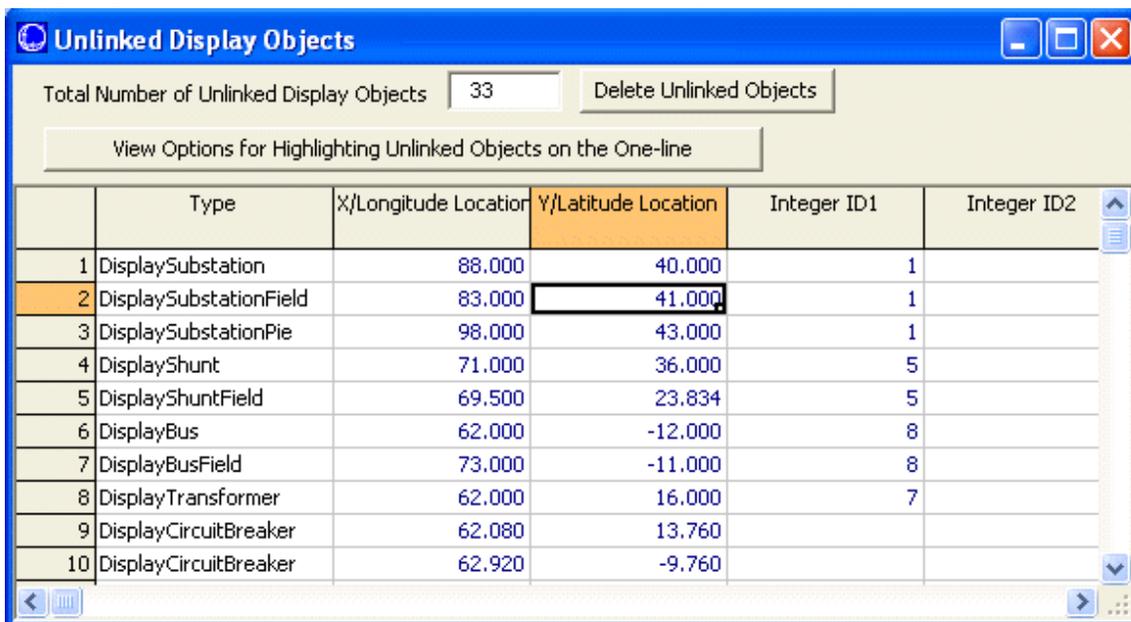
Assume you would like to create an equivalent containing all the buses in areas 1-5 and 10, plus any tie buses, and bus number 2050.

- Since initially all buses are in the study system, first enter 1-1000 in the **Areas** field of **Add to External System**. Since the area of every bus is within this range, this places all the buses in the external system. Alternatively, click **Set All as External** to accomplish the same objective.
- Set **Include how many tiers of neighbors?** to 1. This indicates that all subsequent selections will affect the specified buses and their first tier of neighbors.
- In the **Areas** field of **Add to Study System**, enter 1-5,10. This places all the buses in these areas, plus any tie buses (since Neighbor Tiers is 1), into the study system.
- In the **Buses** list, double-click in the **Which System** field for bus 2050 to change its status from *External* back to *Study*.

List Unlinked Display Objects

An unlinked object is a display object not linked to a record in the power system model (model object). The existence of unlinked display objects on a oneline diagram can be misleading because they have zero flows and zero bus voltages associated with them. See [Object Relationships](#) for more information.

Go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **List Display > Unlinked Display Objects** from the main menu in Edit Mode to identify unlinked objects.



Unlinked Display Objects Dialog

Total Number of Unlinked Display Objects

Indicates the total number of unlinked objects on the display. Ideally, this number should be zero. If nonzero, the unlinked objects are identified in the table by type, screen location, and zoom range over which the object is visible.

Delete Unlinked Objects

Click on this button to permanently remove all unlinked objects from the display. Exercise this option carefully. Generally, you will want to do this either when you have substantially modified a power flow case, such as by creating an equivalent, or when you are using the oneline with a new power system case.

View Options for Highlighting Unlinked Objects on the Oneline

Click on this button will open the [Grid/Highlight Unlinked Objects](#) tab of the [Oneline Display Options Dialog](#).

Type, X/Y Location

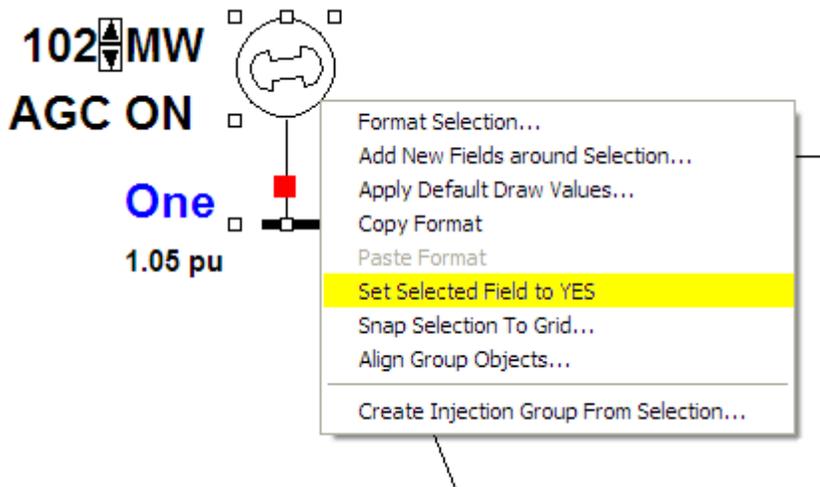
The remainder of the display shows the type, location, identification, layer, applicable zoom level, anchored property, and font size for each unlinked object. This table is a type of [Case Information Display](#) and thus behaves similarly to all other case information displays. Right-click on a record in this table to invoke the local menu. Select **Pan to Object on Open Onelines** to locate and select the unlinked object on the oneline diagram.

Set Selected Field

The Set Selected Field option is an often unused featured in Simulator, but can at times prove very useful. Each object in Simulator has a property called **Selected?**, which does nothing in Simulator other than allow the user to choose a specialized set of objects for some other purpose, such as [advanced filtering](#) a display. The user can add a column to most types of [case information displays](#) which shows the value of the **Selected** property for the type of object being observed. By default, this field is always set to *No*. However, the user can change the value of this field, and then sort the column, filter the display, or any other action that can normally be performed on a column of a Yes/No type.

Set Selected Field to YES from a Oneline diagram

If you right-click on a group of selected oneline objects, one of the local menu options that appears is Set Selected Field to YES. Choose this option to change the Selected field for all the underlying data objects' Selected field to YES.



Set Selected Field for Network Cut

In addition to modifying the **Selected** property manually in a [case information display](#), it is also possible to define a group of buses' **Selected** property by defining a [network cut](#) on the system. Choosing the **Set Selected Field for Network Cut** option from the **Tools** menu will open the [network cut](#) dialog automatically for setting the **Selected** property of a group of objects according to a desired cut plane chosen.

Network Cut

The Network Cut tool is another method for choosing sets of buses for such features as [Scaling](#) or [Equivalencing](#).

The use of the network cut method is to define a set of branches as the "cut" plane, then choose a bus on one side or the other of the cut to indicate which side of the cut you are interested in. The network cut dialog provides the functionality necessary for defining the network cut.

Defining Network Cut

The first step in defining the network cut is to choose the branches which define the cut plane in the system. The one caution is to be sure you select a closed loop cut plane. In other words, you must select a set of branches which completely topologically separates two portions of the system. If you only choose a partial cut plane, which does not completely cut the system in two distinct pieces, then attempted use of the ill-defined cut plane will fail.

To choose the branches which define the cut plane, simply highlight the branches by clicking on them in the list. For each branch you highlight, you must click on arrow button (pointing to the box on the right) to add the branch to your cut plane definition. You can make use of the control (Ctrl) and shift keys to select multiple branches at one time in the list of branches. Note that you can also select a cut plane similarly using interfaces or DC lines.

Once you have the branches (or interfaces or DC lines) selected which form the cut plane, you then need to choose a bus on either side of the cut plane to indicate which side you are interested in. The bottom panel on the display allows you to locate and select this bus.

Require Paths to be Energized

This option can be checked if the branches forming a network cut should be energized to be included.

Include How Many Tiers of Neighbors

Once the network cut has been defined by a set of branches and the bus defining which side of the cut is being examined has been chosen, this option will then include buses within so many tiers of the network cut boundary, on the opposite side of the cut as the specified bus. If the number of tiers is set to zero, then the buses examined will only be those on the same side of the cut as the bus selected.

Setting the Field

Once the network cut has been defined and the side of the cut to be examined has been chosen, the appropriate field for each bus in the area of interest can be set. If the network cut is for the equivalencing tool, then the field to set is the **Which System** property, which defines if a bus is in the external or study system when creating an equivalent. If the network cut is for the scaling tool, then the field to set is the **Scale** property, which defines if a bus is to be included in a scaling action.

Filter by KV

The buses within the network cut can be filtered by nominal voltage level before the field(s) are set. Only buses that are within the network cut and are within the nominal KV level specified will have their fields set to the desired value.

Use Area/Zone Filters

The buses within the network cut can be filtered by the area or zone in which they are contained before the field(s) are set. Only buses that are within the network cut and whose area meets the defined Area/Zone filter specified will have their fields set to the desired value.

Browse Open Onelines

All the objects in all the open .pwd files can be listed in the Browse Oneline Environment dialog. To do so, go to the [Onelines](#) ribbon tab, [Active](#) ribbon group and choose **List Display > Browse open onelines** in Edit Mode or Run Mode. The power system objects are listed on tabs according to their type. For instance, all the buses and bus fields are listed in the **Buses** tab while all the generators and generator fields are listed in the **Generators** tab. The **Others** tab lists all the objects on the onelines which are not associated with power system elements, e.g. background lines or images. The **All** tab lists all the objects in all the open onelines. The default display columns for each grid include the oneline file name of the object and the type of the object. Objects that are linked to case elements also have identifying information about the data element that they represent. The **Others** and **All** pages include the default display columns for generic display objects.

The oneline displays can be panned to the selected object. To do so, select one object from the Browse Oneline Environment dialog and right click to invoke the local popup menu. Click **ScreenObject Records > Pan to Object on Open Onelines**. This will bring the oneline with the selected object to the front and pan to the selected object. When panning, the browsing dialog can be kept on top if **Keep browsing dialog on the top when choosing Pan Oneline to Object** is checked. Otherwise, the dialog will be sent to back when panning.

Unused Bus Numbers

When making changes to a power system case, it is often beneficial to know which bus numbers are still available. To save a list of unused bus numbers to a text file, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Modify Case > List of Unused Bus Numbers** from the [Edit Mode](#) ribbon group. Enter a [range of bus numbers](#) in the dialog and click **OK**. In the Save As Dialog, specify the text file where the list of bus numbers should be written and click **Save**. Only those unused bus numbers in the specified range will be listed in the file. Clicking **Cancel** at any point will abort saving the list to a file.

Palette Overview

The display object palettes are designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. Display object palettes exist for [Areas](#), [Buses](#), [Substations](#) and [Zones](#). Palettes list the display objects that you have already added to the oneline, the display objects that have not yet been drawn that neighbor those displayed objects, and the set of all display objects that have not yet been added to the drawing. By selecting and dragging a display object name from either the **Undisplayed Neighbors** list or the **All Undisplayed** list to a location on the oneline diagram where you would like that object to appear, you can add that display object to the drawing with very little effort.

To see which elements neighboring a particular device are already on the oneline diagram and which are not, highlight a device in either the **Displayed** or **All Undisplayed** list. The **Displayed Neighbors** and **Undisplayed Neighbors** columns will list the corresponding neighboring devices for the selected device, allowing you to drag the undisplayed neighbors to the diagram if you wish.

The display object palettes, in conjunction with the of other devices, are especially useful for adding a large region of an interconnection in relatively little time.

To display one of the four available [palettes](#), select **Show Insert Palette For** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. To display the palette with the focus set to an object already displayed, right-click on that object on the diagram and select ***** Palette** (where *** is the name of the object type) from the popup menu.

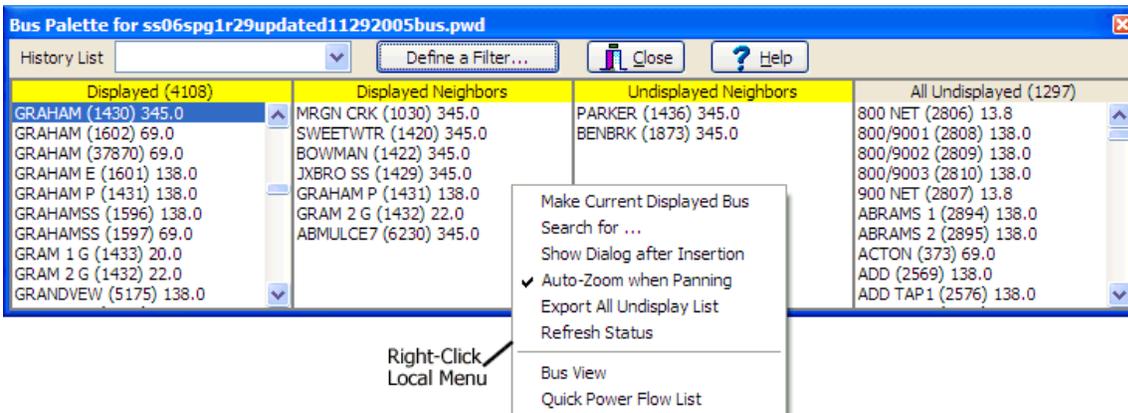
Using the Insert Palettes



The display object palettes are designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. Display object palettes exist for [Areas](#), [Buses](#), [Substations](#) and [Zones](#). To open one of the palettes you must be in [Edit Mode](#) and then go to the [Draw](#) ribbon tab, and choose the **Show Insert Palette For Menu** from the [Quick Insert](#) ribbon group.

The Insert Palettes can be used in conjunction with the various auto-insert capabilities also found on the [Quick Insert](#) ribbon group. These features allow for the creation of oneline diagrams much more quickly than inserting oneline objects one at a time using the [Individual Insert](#) ribbon group.

When you choose one of the palettes the dialog which appears will look like the image below. The following image shows the Bus Palette.



Palettes list the display objects that you have already added to the oneline, the display objects that have not yet been drawn that neighbor those displayed objects, and the set of all display objects that have not yet been added to the drawing.

The display object palettes feature the following controls:

Displayed

Lists those objects defined in the power flow case that have already been added to the oneline diagram. When you click on an entry in the **Displayed** list, the contents of the **Displayed Neighbors** and **Undisplayed Neighbors** lists will be refreshed to identify all objects that neighbor the display object you selected. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Double click on an entry in this list to pan to it so that the selected display object appears in the center of the screen. If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View](#) to learn more about the selected bus.

Displayed Neighbors

Lists those objects defined in the power flow case that neighbor the display object selected in the **Displayed** list that have already been drawn on the oneline diagram. You can make a "Displayed

Neighbor" the selected "Displayed" object by right-clicking on a display object in the **Displayed Neighbor List** and selecting **Make Current Displayed** from the local menu. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Undisplayed Neighbors

Lists those display objects defined in the power flow case that neighbor the object selected in the **Displayed** list that have not yet been drawn. To add the undrawn display object to the oneline diagram, select its name with the left mouse button and keep the left mouse button pressed as you move the mouse to the point on the oneline where you would like to drop the object. When you let go of the left mouse button, the **Information Dialog** box for that display object will appear. (If you do not want the information Dialog box to appear the use the right-click local menu to uncheck the option to regarding showing the information dialog). Use the Information Dialog to change display parameters for the object and click **OK** to finish dropping the object onto the oneline. The display object you have just added will be appended to the end of the **Displayed** list and will also be added to the **History List** so that you can identify its own undrawn neighbors quickly. Only the display objects matching the type of Insert Palette opened will be shown in the list.

If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View Display](#) to learn more about the selected bus.

All Undisplayed

Lists those display objects defined in the power flow case that have not yet been added to the oneline diagram, regardless of whether the object neighbors a displayed object or not. This list functions identically to the **Undisplayed Neighbors List**. To add the undrawn display object to the oneline diagram, select its name with the left mouse button and keep the left mouse button pressed as you move the mouse to the point on the oneline where you would like to drop the object. When you let go of the left mouse button, the **Information Dialog** box for that display object will appear. (If you do not want the information Dialog box to appear the use the right-click local menu to uncheck the option to regarding showing the information dialog). Use the Information Dialog to change display parameters for the object and click **OK** to finish dropping the object onto the oneline. The display object you have just added will be appended to the end of the **Displayed** list and will also be added to the **History List** so that you can identify its undrawn neighbors quickly. Only the display objects matching the type of Insert Palette opened will be shown in the list.

If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View Display](#) to learn more about the selected bus.

History List

Identifies the display objects you have added to the oneline diagram using the insert palettes since the last time you opened the display object palette. Select a name from this list to display the undrawn neighbors of the corresponding display object. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Define a Filter

The **Define a Filter** button opens the [Advanced Filters](#) dialog. This dialog allows you to customize which display objects appear in the various lists. This can be helpful, for example, if you wish to add objects to the oneline that reside only in particular areas, or if you don't want objects less than a certain voltage level to be listed.

When you click **OK** on the [Advanced Filters](#) dialog, the display object palette's lists will automatically be updated to reflect the filter settings.

Close

Click **Close** when you are done using the display object palette.

Right-Click Local Menu of the Palette

There are several options available on the Right-Click

Search for : Click this option to search through the lists for a particular object. It will open the familiar [Find Object dialog](#).

Auto-Zoom When Panning : When double-clicking on the displayed objects in the palette, you will automatically be panned so that object is in the center of the screen. Check this option to also automatically zoom in/out to a reasonable level at the same time.

Export All Undisplayed List : Choose this option to export this list of undisplayed objects to a text file.

Refresh Status : Choose this option to ensure that the list of Displayed/Undisplayed objects is correct.

Show Dialog after Inserting : check or uncheck this option to control whether the information dialog for the object will appear when using the palette

Automatically Inserting Buses

Simulator can automatically insert buses on your oneline diagram if you have data regarding their spatial or geographic location. To achieve this, go to the [Draw](#) ribbon tab and choose **Auto Insert > Buses** on the [Quick Insert](#) ribbon group. This opens the Auto Insert Bus Dialog.

Note: Auto-insert buses is only for inserting bus objects on the diagram representing existing data. In other words, you cannot use the auto-insert buses routine to add new buses to the load flow model. See topics on loading data from [auxiliary files](#) or from [Excel](#) for creating new buses (or other objects) in an existing load flow case.

Insert by longitude, latitude

If you have stored latitude and longitude information with the Bus records in Simulator, you can use that information to automatically insert the buses on a oneline diagram. Select the option on the dialog for finding bus locations based on **Longitude and Latitude stored with data records**. When selected, the [Map Projection](#) option will become enabled. Choose the type of projection you would like to be used for placing the buses on the diagram.

Insert by locations specified in a file

You must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates or longitude, latitude coordinates. The format of the location data text file is as follows:

The first line of this text file is ignored by Simulator. The following lines consist of three numbers: Bus Number, X location, and Y location. If you are reading longitude, latitude, then X signifies longitude and Y signifies latitude.

Num, X Location, Y Location

1, 24001.46, 19715.15

3, 24001.46, 19715.15

16, 24130.91, 19638.99

17, 24007.31, 19093.09

21, 23649.27, 18439.07

22, 23649.27, 18439.07

etc...

Simulator will place the buses on the oneline diagram at the X, Y locations given. If you specified that the file contained longitude, latitude information, then select the [Map Projection](#) to use when converting the longitude, latitude values from the file. The buses will be drawn according to default bus object information defined in the Default Drawing Values for New Objects dialog.

If you would like Simulator to **Autoinsert transmission lines when finished** with the auto insertion of your buses, check the box. Transmission lines and transformers will be automatically drawn based on the Line/XFMR default options defined in the [Default Drawing Values for New Objects](#) dialog.

Check the **Insert bus only if not already shown on the oneline** to ensure that multiple display objects are not inserted for the same element.

Clicking the **OK** button will instruct Simulator to continue by placing the buses according to the specified settings on the dialog. Click **Cancel** to exit the process without inserting the buses.

Automatically Inserting Transmission Lines

The Automatic Insertion of Lines/Transformers Dialog is used to automatically draw transmission lines and transformers on the oneline diagram between existing bus display objects. Only branches that are already defined in the power flow case can be added automatically; if you need to define a brand new branch, see [Transmission Line Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert lines and transformers automatically, you must first have drawn the buses for each end of the device. Simulator then draws the branch display objects connecting the buses for each transmission line/transformer in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Lines** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw line and transformer display objects between terminals whose nominal voltages meet or exceed the minimum kV level specified in this field. For a branch object to be drawn, either one of its terminals must satisfy this criterion. This option is useful for suppressing the automatic addition of generator step-up transformers if that kind of detail is not warranted.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of transmission line stubs, text fields, circuit breakers, and pie charts are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

If this box is checked, the default fields associated with the transmission line will appear around the transmission line.

Insert Equivalenced Objects

If this option is checked, [equivalenced](#) objects modeled as lines will also be automatically inserted with the real transmission lines.

Use Only Selected Buses

Instead of having Simulator automatically insert line and transformer display objects throughout the oneline diagram, you can force it to insert the new objects only between the [bus display objects](#) that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Insert Pie Chart for Lines with No Limit and Bus Ties

If this option is checked, Simulator will add [pie chart objects](#) to the lines that have no given limit or are bus ties as they are auto-inserted. Typically if a line is a bus tie or has no given limit, it is meaningless to include a pie chart on the element, since no relevant information about the transmission element can be gained from the pie chart object.

Insert Multi Section Lines

Check this option if you wish for Simulator to automatically insert lines designated as [multi-section lines](#) as well.

Identifying Bus Ties

This area deals with lines used as ties between breakers. These lines are modeled as zero impedance connections. The identification of a branch as a bus tie depends on the value specified as the **Maximum P.U. Impedance for Bus Ties**. Branches with total P.U. impedance below this value will be considered bus ties when auto-inserted.

In this area two choices are given for how to insert the bus tie breakers: **Do not insert stubs for bus ties** and **Only insert a single circuit breaker**. The first choice allows you to decide if line stubs will be drawn when the tie breaker is inserted. The second sets whether or not only one circuit breaker is inserted on the tie breaker. This could be useful for determining real lines from bus tie breakers.

Automatically Inserting Generators

The Automatic Insertion of Generators Dialog is used to automatically draw generators on the oneline diagram on existing bus-display objects. Only generators that are already defined in the power flow case can be added automatically; if you need to define a new generator, see [Generator Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert generators automatically, you must first have drawn the terminal bus for each device. Simulator then draws the generator display objects connected to the buses for each generator in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Generators** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw generator display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a generator object to be drawn, its terminal must satisfy this criterion.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

When this box is checked, the default fields associated with the generator will appear around the generator, such as voltage, name, and/or MW.

Insert Equivalenced Objects

This field is not used when automatically inserting generators.

Use Only Selected Buses

Instead of having Simulator automatically insert generator display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Automatically Inserting Loads

The Automatic Insertion of Loads Dialog is used to automatically draw loads on the oneline diagram on existing bus display objects. Only loads that are already defined in the power flow case can be added automatically; if you need to define a brand new load, see [Load Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert loads automatically, you must first have drawn the terminal bus for each device. Simulator then draws the load display objects connected to the buses for each load in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Loads** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw load display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a load object to be drawn, its terminal must satisfy this criterion.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

When this box is checked, the default fields associated with the load will appear with the loads, such as MVAR and/or MW.

Insert Equivalenced Objects

If this option is checked, [equivalenced](#) objects modeled as loads will also be automatically inserted with the real transmission lines.

Use Only Selected Buses

Instead of having Simulator automatically insert load display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Automatically Inserting Switched Shunts

The Automatic Insertion of Switched Shunts Dialog is used to automatically draw switched shunts on the oneline diagram on existing bus-display objects. Only switched shunts that are already defined in the power flow case can be added automatically; if you need to define a brand new switched shunt, see [Switched Shunt Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert switched shunts automatically, you must first have drawn the terminal bus for each device. Simulator then draws the switched shunt display objects connected to the buses for each switched shunt in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Switched Shunts** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw switched shunt display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a switched shunt object to be drawn, its terminal must satisfy this criterion.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

When this box is checked, the default fields associated with the switched shunt will appear around the switched shunt, such as nominal MVAR.

Use Only Selected Buses

Instead of having Simulator automatically insert switched shunt display objects throughout the oneline diagram, you can force it to insert the new objects only at the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Automatically Inserting Interface Display Objects

The Automatic Insertion of Interfaces Dialog is used to automatically insert [interface objects](#) on the oneline diagram between existing [area/zone display objects](#). [Interface pie chart objects](#) can also be inserted as part of this process. The automatic insertion of interface display objects greatly accelerates the construction of interface diagrams, which are particularly useful for animating the results of [PTDF](#) calculations. Area-area and zone-zone interface records can be displayed using interface display objects as well as interfaces comprised only of transmission branches.

Note that inserting interface objects on a oneline diagram does NOT add interface objects to the load flow case. The interface definitions need to be added to the load flow case prior to automatically inserting the graphical interface objects on a oneline diagram. The quickest way to add area to area or zone to zone interface definitions in the load flow case is to [automatically insert the definitions](#) in the interface case information display.

Once you have the interface definitions defined in the case, you can automatically insert the interface objects on a diagram using the following procedure:

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **New Oneline** from the [application button](#)), place [area/ zone objects](#) at the desired locations if inserting area-area or zone-zone interface records. Otherwise, the terminal buses for the branch elements in the interface should be inserted at the desired locations on the oneline diagram.
- Go to the [Draw](#) ribbon tab and choose **Auto Insert > Interfaces** on the [Quick Insert](#) ribbon group.
- Check the **Insert Pie Charts on Interfaces** box to automatically insert interface pie chart objects when the interfaces are inserted. If this option is selected, change the **Default Size of Interface Pie Charts** to specify their size.
- Choose the type of interfaces to insert. **Area to Area or Zone to Zone Interfaces** and **Line/Transformer Interfaces** can be inserted. Area-area and zone-zone interfaces will be drawn between the respective area and zone objects displayed on the oneline. Line/transformer interfaces will be drawn based on the average location of the terminal buses of all transmission lines comprising the interface. If inserting line/transformer interfaces, the option **Minimum Length of Line/Transformer Interfaces** can be set to prevent interfaces that are too short from being inserted.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted based on the selected criteria for any corresponding interface record that is not already represented.

Note that you can do this automatic insertion as often as you like. The **Number of Interfaces Not Shown** field indicates how many interfaces still need to be added to the diagram to represent all defined area-area, zone-zone, or line/transformer interfaces. It is a read-only field.

Automatically Inserting Substations

Simulator can automatically insert substations on your oneline diagram if you have data regarding their spatial or geographic location. To achieve this, go to the [Draw](#) ribbon tab and choose **Auto Insert > Substations** on the [Quick Insert](#) ribbon group. This opens the Auto Insert Substation Dialog.

Insert by longitude, latitude

If you have stored latitude and longitude information with the Substation records in Simulator, you can use that information to automatically insert the substations on a oneline diagram. Select the option on the dialog for finding substation locations based on **Longitude and Latitude stored with data records**. When selected, the [Map Projection](#) option will become enabled. Choose what type of projection you would like to be used for placing the substations on the diagram.

Insert by locations specified in a file

You must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates or longitude, latitude coordinates. The format of the location data text file is as follows

The first line of this text file is ignored by Simulator. The following lines consist of three numbers: Substation Number, X location, and Y location. If you are reading longitude, latitude then X signifies longitude and Y signifies latitude.

Num, X Location, Y Location

1, 24001.46, 19715.15

3, 24001.46, 19715.15

16, 24130.91, 19638.99

17, 24007.31, 19093.09

21, 23649.27, 18439.07

22, 23649.27, 18439.07

etc...

Simulator will place the substations on the oneline diagram at the X, Y locations. If you specified that the file contained longitude, latitude information, then select the [Map Projection](#) you would like to be used when converting the longitude, latitude values from the file. The substations will be drawn according to default substations object information defined in the Default Drawing Values for New Objects dialog.

Finally, if you would like Simulator to **Auto insert transmission lines when finished** with the auto insertion of your substations, check this box. Transmission lines and transformers will be automatically drawn based on the Line/XFMR default options defined in the [Default Drawing Values for New Objects](#) dialog.

Check the **Insert substation only if not already shown on the oneline** to ensure that multiple display objects are not inserted for the same element.

Clicking the **OK** button will instruct Simulator to continue by placing the substations according to the specified settings on the dialog. Click **Cancel** to exit the process without inserting the substations.

Automatically Inserting Borders

PowerWorld Simulator allows you to automatically insert geographic borders, which PowerWorld Corporation has drawn. These include the states in the United States of America and several international borders as well. You may also define a border in a text file and insert this User-Defined border. It is highly recommended that you create a diagram and insert borders before you begin adding power system objects to the diagram.

To bring up the **Auto Insert Borders Dialog**, go to the [Draw](#) ribbon tab and choose **Auto Insert > Borders** on the [Quick Insert](#) ribbon group.

Options

The Options tab allows you to set characteristics of the border lines when they are inserted on the oneline diagram. Most of these options are available from the Format menu in Edit Mode, and can be changed for a border by selecting the border(s) in Edit Mode and then selecting the appropriate option from the Format menu to change the desired option(s).

Make border lines immobile

This option is very useful for preventing you from inadvertently moving the border lines in Edit Mode once they have been placed.

Automatically link county and state borders to supplemental data

Background lines created from inserting borders can be linked to [Supplemental Data](#) records for purposes of further identifying these objects. This is useful when applying filters, applying dynamic formatting, using Select By Criteria, or defining custom hints with display objects created from inserting the borders. This option is only available when inserting United States borders.

When this option is checked, background lines created for the borders will be automatically linked to Supplemental Data records. If necessary, Supplemental Data records will be created with the Classification being either *State* or *County*, depending on the Border Type selected when inserting United States borders, and the Name being the name of either the state or county.

Line Options

Choose the line thickness and line color for the border lines.

Fill Options

Choose the fill options if you wish to "fill in" the border regions with a background fill color.

Stack Level

Choose the stack level for the border lines. The stack level affects which objects appear "over" or "under" other objects on the diagram. Typically background lines are either placed in the Background or Base levels, so that they appear beneath power system objects on the diagram. Further we would recommend that if you are using background fill colors for the borders that you choose a stack level of Base. The only difference between the Background and Base levels is that the Base level objects can be right-clicked on in Run Mode and allows the default diagram popup menu to appear. Right-clicking on Background level objects in Run Mode will not display the default popup menu.

Layers

When Borders are automatically inserted on a diagram using this tool, a [Borders screen layer](#) will be added to the layer drop-down box, if a *Borders* layer does not already exist for the oneline diagram.

You can choose to leave the borders in the standard *Default* layer, to place the borders in the new *Borders* layer, or you can [create a new custom layer](#) by clicking the **Define Layers** button. If no default borders layer has been previously established (borders were previously inserted on the oneline), the *Borders* layer or selected user created layer will be set as the default Borders layer. The default Borders layer is different than the standard *Default* layer for objects. The default Borders layer information is stored in the system registry, and is only used to recall which layer borders were assigned to the previous time borders were automatically inserted.

Apply Default Drawing Values

Click this button to apply the Background line settings from the [Default Drawing Values](#).

Border File Path

Designate the border file location.

PowerWorld Library

Pre-Defined Selections

Choose between North America, USA State Borders, Canadian Province Borders, or Entire World to insert the selected region. These options make things quicker than selecting from the individual menus if entire countries or continents are to be inserted.

Once you have selected the regions that you want to insert, Click **OK**.

United States

This tab allows you to select states in the United States that you want to insert. (Note: to select several states to insert at once, use the Ctrl and Shift keys while clicking with your mouse.) These states will be placed on the screen such that as you add new states they will be placed geographically appropriately.

Once you have selected the states you want to insert, Click **OK**.

Canada

This tab allows you to select provinces in Canada that you want to insert. (Note: to select several provinces to insert at once, use the Ctrl and Shift keys while clicking with your mouse.) These provinces will be placed on the screen such that as you add new provinces they will be placed geographically appropriately.

Once you have selected the provinces you want to insert, Click **OK**.

World

This tab allows to insert borders which PowerWorld Corporation has acquired for countries around the world. PowerWorld Corporation will continue to add more border files as we receive them. Note: if you have drawn a oneline which contains a geographic border you would like us to include in future version of Simulator, please contact us at support@powerworld.com and we'll add it to our next release.

Map Projection

This setting is important for anyone including border files outside of North America. By default, Simulator draws borders using a simple conic projection referenced to North America. This can result in border files of other countries being drawn incorrectly if based on this reference. To correctly draw borders of other countries around the world, select the Entire World setting, which uses a Mercator projection.

User-Defined Borders

This tab allows you to read in a border from a text file that you create. The first row of this text file is a comment row and is ignored when reading it in. After this it reads in the description of each background line. This description starts with the number of points in the line followed by a list of x , y coordinates. The file ends when the number of points for a line is read as -1. An example file follows:

Comment row

5

59, 60

10, 20

40.3, 95.20

89.3, 22.11

79.5, 34.56

45

40, 66

etc...

-1 this signifies the end of the file

To read the data, first specify the file name and location containing the data using the Browse button. Next indicate if the file contains data in Longitude, Latitude or in converted Simulator X,Y coordinates. If the data is Longitude and Latitude, then also specify if the map projection to use should be North America (simple conic) or Entire World (Mercator).

Equivalents

An equivalent power system is a power system model of smaller dimension than the original system that approximates the behavior of the original system reasonably well. In reality, most power system models are actually an "equivalent" of a much larger interconnected network. When performing power system studies, it may be desirable to reduce the size of the system model even further so that it may be solved more quickly.

To bring up the [Equivalents Display](#), while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Create Equivalent** from the [Edit Mode](#) ribbon group.

The most important part of constructing an equivalent is determining which buses should be explicitly retained in the equivalent, and which buses should be *equivalenced*, or removed from the case. Several definitions are useful here:

Study System

The buses that are to be retained.

External System

The buses that are to be equivalenced.

Boundary Buses

Any buses in the study system that are connected to buses in the external system.

How well the equivalent system approximates the behavior of the original system depends upon which buses are retained in the study system. Retaining more buses yields results that more closely match those of the original case, but at the expense of greater computation time. The number of buses to retain in the study system depends upon how the equivalenced system will be used. Building system equivalents is as much an art as it is a science, with few solid rules of thumb. However, to improve accuracy, you should retain as many generator units as possible.

The actual equivalent is constructed by performing a matrix reduction on the bus admittance matrix. A result of this process is the creation of "equivalent" transmission lines that join boundary buses equipped with equivalent shunts or loads. Equivalent lines typically have a circuit identifier of *99*, but have also been seen to have other numerical values between 90 and 99, or an alphanumeric identifier of *EQ*. Since many of the equivalent lines created during the matrix reduction have very high impedance values, an option is provided to ignore equivalent lines with impedances exceeding a specified threshold value. Additionally, an option is provided to convert the equivalent shunts added at the boundary buses to constant PQ loads. These PQ loads will be given circuit identifiers similar to those given to equivalent transmission lines.

Equivalents Display

The **Equivalents Display** is used to construct equivalent systems. An *Equivalent System* is a system of smaller dimension that exhibits similar power flow properties. Equivalent systems are constructed to help accelerate computation time without sacrificing a significant amount of accuracy. For more information, please see [Equivalents](#).

To bring up the Equivalents Display, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Create Equivalent** from the [Edit Mode](#) ribbon group. This display contains two pages, the **Bus Selection Page** and the **Create Equivalent Page**. Use the Bus Selection Page to partition the power system into the study system and the external system. Use the Create Equivalent Page to

- Save the external system in a file
- Extract the external system
- Build an Equivalent

Each of these tasks is described below.

Bus Selection Page

To perform any of the tasks described on the Create Equivalent Page, you first need to specify the study system and the external system. Do this by directly assigning buses to the desired system. The [Bus Selection Page](#) has been designed to provide a number of powerful and complimentary ways of accomplishing this task.

The most important point to keep in mind when using this page is that membership in the study system and the external system is on a bus-by-bus basis (as opposed to by areas or zones). Thus, each bus is either in the study system or the equivalent system. Each bus' current assignment is indicated in the Buses list, which is shown on the bottom left corner of the page. The Buses list is a [Case Information display](#) and can be manipulated similar to other displays. By default, all buses initially reside in the study system. Please see [Bus Selection Page](#) for more details.

Create Equivalent Page

The Create Equivalent Page allows you to save the external system in a file, to delete the external system, and to build the power system equivalent.

Build Equivalent

This procedure constructs an equivalent system. For background on equivalents, please see [Equivalents](#). The following sections contain the options for building an equivalent.

Delete All External Generators or Retain Generators with Max MW Ratings Above

If the **Delete All External Generators** checkbox is checked, the equivalencing routine will remove all external generators from the case, regardless of their MW rating. Otherwise, the equivalencing routine will add to the study system any generators originally in the external system whose real power output exceeds the specified value. Retaining large generators often makes an equivalent significantly more accurate. If you do not wish to retain any additional generators, check the **Delete All External Generators** checkbox.

Retain Remotely Regulated Buses

Some generators and transformers regulate buses other than their terminals. When this box is checked, these remotely regulated buses are automatically included in the equivalent if the regulating generator or transformer is included. If the box is not checked, the regulated bus is set to the terminal of the retained object. It is strongly recommended that you leave this box checked at all times.

Retain Branch Terminals For

This section allows you to customize the retention of branch terminals for special-case types of branches. You may choose to retain terminal buses for transformers, zero impedance ties, area tie lines, and/or zone tie lines.

Max Per Unit Impedance for Equivalent Lines

During the equivalencing process, a number of equivalent lines are created joining the boundary buses. All equivalent lines with per unit impedance values above this threshold are ignored.

Two Character Circuit ID for New Equivalent Lines

Choose the circuit identifier to be used for the equivalent lines that are created. Typical circuit IDs for equivalent lines are *99* or *EQ*.

Remove External Objects from Onelines

This feature removes display objects associated with the external system from any open onelines.

Convert Equivalent Shunts to PQ Loads

During the equivalencing process, shunt elements are added at the boundary buses. Check this box if you would like these equivalent shunts converted to constant PQ loads. If this option is checked, equivalent loads are created with a [load ID](#) of *99*.

Remove Radial Systems

Checking this option results in all radial connections in the network to be reduced to their nearest non-radial bus (i.e. node.) The equivalencing routine will iteratively reduce the network when this option is checked, until no more radial connections exist in the system.

Delete Empty Areas/Zones/Substations that occur from Equivalencing

Since equivalencing is a process which ultimately removes buses from the system, and Areas, Zones and Substations are system devices which are groups of buses, this option will automatically remove the definition of these types of objects when all buses within them are removed from the case during the creation of the equivalent.

Adjust Area Unspecified Interchange to Zero Out ACE

Selecting this option will allow Simulator to automatically change the unspecified MW interchange amounts of each area to remove any non-zero ACE discrepancies caused by the equivalencing. This will help prevent these discrepancies from being assigned to the system slack bus.

Select **Build Equivalent System** to construct the equivalent system. Constructing an equivalent system permanently removes the external system from the case and adds a number of equivalent lines and shunts/loads.

Saving the External System in a File

This procedure allows an external system to be saved in a file **without deleting** the external system. This option is useful for allowing you to save a portion of the system in a file, modify it using perhaps another program, and then use [Append Case](#) to append the modified file to the original case.

When saving the external case, there are two options: 1) save just the external case, or 2) save the external case and any ties to the original case. Option one just saves the external case, while option two saves the external case and any transmission lines or transformers that connect the external system to the rest of the system. Save just the external case if you are planning to use the external case as a standalone case. Save the external case and its ties if you are planning to modify the external case and then to append it back to the original case.

Click **Save External System** to save the external system in a file. You will be prompted for the desired [case format](#).

Deleting the External System

This procedure deletes the external system from the original case. All devices in the external system are removed, including any buses and lines/transformers in the external system and any lines/transformers that join the external system to the study system. Check the **Remove External Objects from Onelines** to remove any display objects linked to the external system from the open onelines.

This command **permanently removes** the external system from the case; an equivalent system is **not** created. Select **Delete External System** to actually delete the system.

Scaling

Use the **Power System Scaling Dialog** to scale the load, generation, or bus shunts uniformly for either the entire case or a group of selected buses. This display allows you to scale any of the following values:

- Bus real power load
- Bus reactive power load
- Generator real power output
- Real component of the bus shunt admittance
- Capacitive component of the bus shunt admittance
- Reactive component of the bus shunt admittance

To display the Power System Scaling Dialog, go to the [Tools](#) ribbon tab, and choose **Scale Case** from the [Other Tools](#) ribbon group. When the dialog appears, you may begin to select the buses to be scaled. Buses can be selected individually or in a group by areas or zones. In addition, if you already have specific groups of devices defined as an injection group, you can choose to scale values associated with the injection group.

Scale by Bus

The button on the left labeled **Bus** enables selection of loads, generators and shunts by the bus to which the devices are attached. Selection of the buses can be done individually, by their area grouping, by their zone grouping, or by their super area grouping. Scaling is done on the buses chosen regardless of the area, zone, or super area specification. Options for areas, zones, and super areas are included only to aid in selecting and unselecting buses to be scaled. This is important to keep in mind if loads or generators are assigned to a different area or zone than their terminal bus.

The **Buses Table** lists the name and number of all buses in the system and whether or not each bus will participate in the scaling. Similarly, the **Areas, Zones, and Super Area** tables list the names and numbers of all areas, zones or super areas in the system and whether or not each area will participate in the scaling. Simulator initially assumes that no buses are to be scaled. The Power System Scaling Dialog furnishes a number of ways to select the subset of buses to be scaled:

- Use the **Add to Scaling** fields to enter either a [range of areas, zones and/or buses](#) to scale.
- Use the **Remove from Scaling** fields to enter either a [range of areas, zones and/or buses](#) to omit from the scaling.
- Use the **Select buses using a network cut** button to define a [Network Cut](#) to choose a custom set of buses within a portion of the system to be scaled or omitted from the scaling.
- Use the **Buses, Areas, Zones and Super Areas** tables to change the scaling status of individual buses. Simply double-click on the **Scale** field for a bus, area, zone or super area to toggle its value.
- Click the **Set All to Yes** button to scale the entire case. Click the **Set All to No** button to remove the entire case from scaling.

Changing the scaling for an area, zone or super area changes the scaling status for all *buses* in the grouping. For example, to scale all the buses in a single area, first click **Set All to No**. Then, click on the **Scale** field for the desired area in the Area Table. To scale all buses in an area except for a select few, repeat the above process, but then click on the **Scale** field for the buses not to scale.

As you select the buses to be scaled, the fields in the **Totals for Selected Buses** are updated to indicate the total load, generation, or shunt compensation that will be scaled.

Once you have selected the buses, you can either use the **Scale By** fields to enter a new scaling factor for each of the quantities or use the **New Value** fields to specify a new value directly. If you do not wish to scale a particular type of device, such as bus shunts, simply leave the **Scale By** field as unity.

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. To scale generation and load to enforce ACE, check **Scale Gen to Keep ACE Constant**. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control. To scale only generators whose AGC field is set to *YES*, check **Scale Only AGCable Generation and Load**. To scale both in-service and out-of-service loads, check **Scale Out-Of-Service Loads**.

Finally, click **Do Scaling** to scale the load, generation, or shunt compensation.

Scale by Area

The button on the left side of the dialog labeled **Area** enables the selection of loads, generators and shunts based on the area designation of the device itself. Loads, generators and shunts can have a different area designation than the terminal bus to which the device is attached. In these cases, it is sometimes necessary to use the **Scale by Area** option to choose only the devices that are designated within a certain area, instead of all devices at a particular bus.

To specify the devices within certain areas to be scaled, toggle the **Scale** property of the areas desired in the Areas table, or use the **Add to Scaling** and **Remove from Scaling** fields to enter [ranges](#) of area numbers to be included or excluded from the scaling.

Scale by Zone

The button on the left side of the dialog labeled **Zone** enables the selection of loads, generators and shunts based on the zone designation of the device itself. This is identical to the concept described immediately above in the discussion on **Scale by Area**. As was the case with the area designation, loads, generators and shunts can have a different zone designation than the terminal bus to which the devices are attached.

To specify the devices within certain zones to be scaled, toggle the **Scale** property of the zones desired in the Areas table, or use the **Add to Scaling** and **Remove from Scaling** fields to enter [ranges](#) of zone numbers to be included or excluded from the scaling.

Scale by Injection Group

If injection groups have been defined for the case, you can scale the generation, load, and switched shunt values for the injection group by first clicking on the **Injection Groups** button. The **Buses Table** will be replaced with a new table listing the Injection Groups in the case. To choose the injection groups to scale, double-click the **Scale** field to toggle the value between *No* and *Yes*. As **Scale** fields are toggled to *Yes*, the **Net Injection** fields will update to display the totals for the selected injection groups. When all of the desired injection groups have been selected, either the **Scale By** or the **New Value** fields can be modified for a desired new value for the selected injection groups. Keep in mind that the scaling is done based on net injection and will be done in proportion to the defined [participation points](#).

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control.

Checking the **Scale in Merit Order** option will scale generators in the order of highest to lowest normalized participation factor. This means that each generator participating in the injection group will be moved to either its maximum or minimum limit, depending on the direction of the scaling, before the output of the next generator in the list is changed, until the total scale amount is achieved. If this option is checked, generator MW limits will be enforced regardless of the status of the **Enforce Gen MW Limits** option.

Check the **Scale Only AGCable Generation and Load** option to scale only those generators and loads whose AGC field is set to *YES*. If the **Ignore AGC flag to calculate participation but use AGC flag to scale individual loads or generators** option is checked, the **Scale Only AGCable Generation and Load** is not enabled and will be ignored. If this option is checked, then all generators and loads, regardless of their AGC status, will be used to calculate the participation percentages for all participating elements. Only those generators and loads whose AGC field is set to *YES* will actually be scaled. Normally, if choosing to scale only AGCable generation and load, only those elements whose AGC status is *YES* will be used to calculate the participation percentages.

The choice of **Scale Starting Point** impacts the reference point from which the scaling starts. **Scale from Present Value** scales by adding an incremental change to the present MW values based on the difference between the **New Value** and **Original Value**. **Scale from Zero** sets all participating elements to zero injection (0 MW load or generation) and then scales to the **New Value** from that starting point. When using the **Scale from Zero** option, only a single injection group can be scaled at a time. The following example explains the difference between these two options:

Suppose that Load 1 is 800 MW and Load 2 is 700 MW. These two loads are the only elements in an injection group and they both have a participation factor of 0.5. The desired **New Value** is 1600 MW. Choosing **Scale from Present Value** will result in Load1 being 850 MW and Load 2 being 750 MW. The incremental change in the total load is 100 MW and both loads pick up half of this change. Choosing **Scale from Zero** will result in Load 1 being 800 MW and Load 2 being 800 MW. Both loads have been scaled to 0 MW initially and they each pick up half of the desired 1600 MW total load.

The **Scale Load Field** options provide a way of scaling load that is out-of-service. Selecting **Actual MW (MW Column)** will scale only those loads that are in-service. (The MW column in the [load case information display](#) reflects the total in-service load at a bus.) Selecting **Modeled MW (ignore load and bus status)** will scale all load regardless of the load and bus status.

Click **Do Scaling** to scale the selected injection groups.

Scale by Owners

You can scale the generation and load values by ownership by first clicking on the **Owners** button. The **Buses Table** will be replaced with a new table listing the Owners in the case. To choose the owners to scale, double-click the **Scale** field to toggle the value between *No* and *Yes*. As **Scale** fields are toggled to *Yes*, the Generator and Load MW and Mvar fields will update to display the totals for the selected owner. When all of the desired owners have been selected, either the **Scale By** or the **New Value** fields can be modified for a desired new value for the selected owners.

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. To scale generation and load to enforce ACE, check **Scale Gen to Keep ACE Constant**. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control. To scale only generators whose AGC field is set to *YES*, check **Scale Only AGCable Generation**. To scale both in-service and out-of-service loads, check **Scale Out-Of-Service Loads**.

Finally, click **Do Scaling** to scale the load and generation for the selected owners.

Note: When scaling generators with multiple owners, the scaling will be done on the entire output of these generators, regardless of the fact that not all of the owners have been set to scale. If this is not a desired behavior, you can define multiple generators at the bus, each with 100% ownership.

Overview of Facility Analysis in PowerWorld Simulator

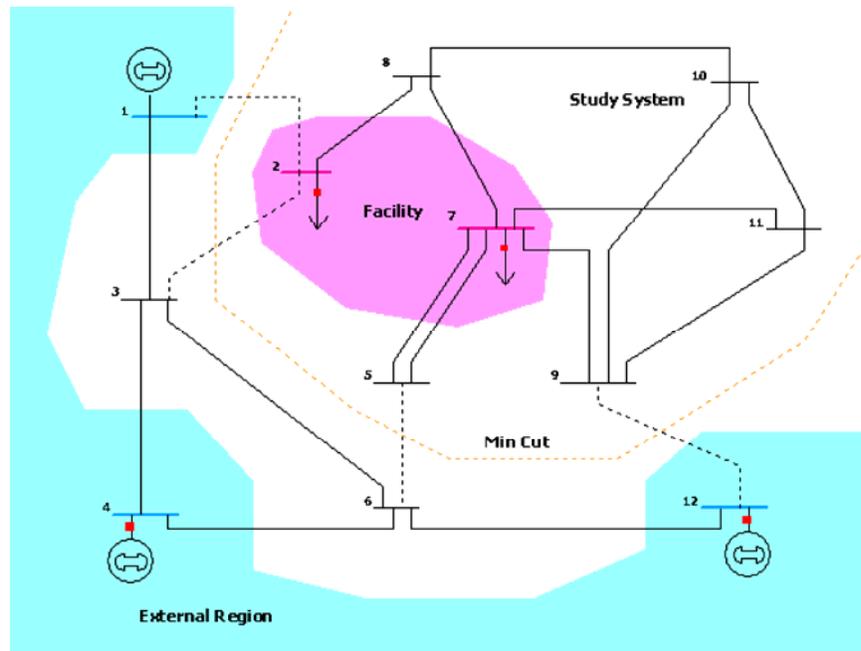
Facility Analysis is used to study the topological redundancy of interconnect-specific electric facilities. This application determines the minimum set of AC transmission lines and transformers that, when opened or removed from the system, would electrically isolate a set of Facility buses from a set of External buses.

The tool is an application of the [augmenting path max flow min cut algorithm](#) with modifications to handle electric networks.

The Facility analysis process has two steps:

1. The [Select the Buses](#) dialog is used to specify the External and Facility buses. Multiple selections of the External buses can be done using any of the area or zone selectors. The Facility buses are specified by setting the **Selected** field of buses to **YES**.
2. The [Facility Analysis dialog](#) is used to determine the Min Cut and visualize the branches that belong to the min cut.

The Facility analysis application runs in Edit Mode and takes into consideration the open or closed status of the branches. Open lines are considered as not present in the system. The figure below illustrates the functionality of the Facility Analysis tool.



Facility Analysis Dialog

This dialog is used to determine the branches that would isolate the Facility from the External region as specified in the [Select the Buses](#) dialog. When switching to the Facility Analysis page, the application builds a graph data structure and reports information regarding the External Region and the Facility.

To access the Facility Analysis Dialog go to the [Tools](#) ribbon tab, and choose **Connections > Facility Analysis Dialog** from the [Other Tools](#) ribbon group.

Select the Buses

The options and use of this page are described in their entirety in the topic titled [Bus Selection Page](#).

Facility Analysis

External Region

The external region is a set of buses from which the Facility would be isolated. Although the buses in the external region may not be adjacent to each other, the algorithm will assume that any of these buses can supply electricity to the Facility and thus the buses in the External region are considered to be connected. The External region is defined using the [Select the Buses](#) dialog.

External Region - Number of Buses

Indicates the number of buses in the External region. This number is equal to the number of buses in the system that were labeled as External in the [Select the Buses](#) dialog. There should be at least one bus in the external region.

External Region - Capacity

Indicates the number of branches that connect the external buses to study buses in the system. This is the outgoing [graph flow](#) capacity from the External region toward the Facility.

Facility

Is the set of buses that constitute the Facility. Although the buses in the Facility may not be adjacent to each other, the algorithm will assume that any of these buses may receive power from the External region, and thus the Facility buses are considered to be connected to each other.

Facility - Number of Buses

Indicates the number of buses in the Facility. This number is equal to the number of buses in the system that were labeled as *Study* and whose **Selected** field is set to *YES*. There should be at least one bus in the Facility.

Note that if a bus was specified to be *External* and its **Selected** field is *YES*, then the application will issue an error, since Facility buses cannot be in the External region. In such cases the status will indicate that the graph structure is incomplete.

Facility - Capacity

Indicates the number of branches that connect Facility buses with study buses. This number is equal to the outgoing [graph flow](#) capacity from the Facility toward the External region.

Status

Shows the status of the Facility Analysis application. If the External region and the Facility are specified, the status will indicate that the graph structure is ready. During execution of the Min Cut algorithm, the status shows the number of the augmenting paths found so far and the number of branches in the current path.

Find Minimum Cut

Press this button to initiate the [Min Cut algorithm](#). The button is inactive if the graph structure is incomplete, i.e., there is either no external bus or facility bus, or facility buses were found inside the External region.

Show Paths

Toggle this button to visualize the augmenting paths at the bottom of the form. The augmenting paths are listed in the order in which they were found. Note that the first path has fewer nodes, since the algorithm uses a shortest (least number of nodes) path routine. For each augmenting path the number of buses in the path, as well as the corresponding bus numbers are listed in the dialog. This button is enabled only if a min cut has been found.

Augmenting Path Max Flow Min Cut Algorithm

The augmenting path max flow – min cut algorithm is used to identify the minimum number of branches that need to be opened or removed from the system in order to isolate the Facility (power system device) from an External region.

The algorithm is an application of the Max Flow - Min Cut theorem, which states that the maximum flow that can be transferred from a set of source nodes to a set of sink nodes across a graph equals the capacity of the minimum cut. The facility analysis in Simulator finds a min cut, although this cut may not be unique.

The application consists of three stages:

1. **Convert the electric network to a graph structure**

In this stage, each branch of the system is converted to an undirected arc with [graph flow](#) capacity equal to one. Thus, only one unit of [graph flow](#) can be sent through a branch. The Facility buses and the External buses are converted to Facility and External supernodes, respectively. This effectively reduces the problem to finding the min cut between these two supernodes. The number of nodes and capacities of the Facility and the External region are also computed during this stage.

2. **Find the Max Flow using the Augmenting Path Algorithm**

This is an iterative process. At each step, the algorithm finds a new augmenting path from the Facility to the External region and augments the [graph flow](#) along this path in one unit. Consequently, the branches in the path won't be available for flow augmentation in the next step. Each new path is determined using a shortest path routine. The algorithm stops when no augmenting path from the Facility to the External region can be found. The number of units transferred from the Facility to the External region (path augmentations) reaches the number of branches in a certain cut. Note that the number of branches in the min cut can not exceed the capacity of either the Facility or the External region.

3. **Determine the branches in the min cut**

The identification of the branches that constitute the min cut consists in tracking down labels in the buses and branches used during each path augmentation.

Graph Flow

Most network and graph theory applications use the concept of flow to represent any object that can be transported, such as communication packets or trucks, but also connectivity properties of graphs. In the [augmenting path max flow min cut algorithm](#) the flow is an artificial concept used to represent topological connectivity of buses. Two buses are adjacent if flow can be sent from one to the other through a branch.

Graph Flow Capacity of a Branch

Networks that transport some flow are said to be capacitated if its arcs (here synonym of "branches") have some limit associated to the flow transportation. For instance, capacity of a communication channel, or number of trucks that can be simultaneously on a certain road. The algorithm used in the Facility Analysis assigns a capacity of one to each branch. This means that the branch can be used only once for "connecting" two nodes.

Graph Flow Capacity of a Cut

The capacity of a topological cut is equal to the sum of the capacities of its arcs. In the Facility Analysis, the capacity of the min cut is equal to the number of branches in the min cut, since each branch has a capacity of one.

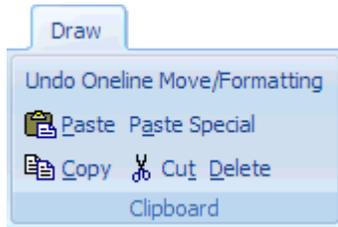
Chapter 6: Editing Online Diagrams

The Format Menu is used in the Edit Mode to customize the appearance of the online diagrams. This chapter covers the following:

- Editing Overview
- Selecting Objects
- Changing Basic Display Object Properties
- Delete, Copy and Undo

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting online objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



Undo

The Undo command is used in the edit mode to undo the last change made on the online diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the online. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

Absolute Coordinates : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different online diagrams which contain the same geographic background.

Coordinates Relative to Cursor : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

Select by Criteria Dialog



To open the Select By Criteria Dialog go to the [Draw](#) ribbon tab and choose **Select By Criteria** from the [Select](#) ribbon group.

The Select By Criteria Dialog provides a way of selecting objects that meet a specific set of criteria. The criteria may include Area Numbers, Zone Numbers, Voltage Levels, Zoom Levels, and Object Type.

Use the dialog's controls to specify the selection criteria. Use the **Area** and **Zone** fields to select the areas and zones in which you want to select display objects. [Ranges of area and zone numbers](#) can be entered in the usual way, or you can check the **All Areas** or **All Zones** boxes to select all areas and all zones. You can also use the **Area** and **Zone** tab pages to check or uncheck areas or zones in which you want to select display objects.

Specify the max and min voltage levels for selected objects using the boxes that are provided, keeping in mind that all voltages are in kV.

Specify the [layers](#) for selected objects in the **Layers** tab. If not all layers are selected, the **Layers Range** option will be selected. If the **All Layers** option is clicked, then all the layers will be selected.

Next, select the type of object in which you are interested from the supplied list. To select items from this list, check the box next to the type of field(s) you desire. Display object types that are followed by a right-arrow, such as *Area Fields*, are expandable, meaning that they have several associated subtypes. If you checked a field type that has subtypes, you can highlight the type in the list to see the available subtype list. You can then more specifically select subtypes that you wish to include. By default, all subtypes of a general object type are selected. You also have the option for the list to display all fields available for a particular object type, or to display only the most commonly selected fields.

You can also choose to associate an [advanced filter](#) with a particular type of object in the list for selecting only objects of that type that meet the filter. To do so, right-click in the Filter column next to the object type for which the filter applies, or click on **Add/Modify Filter...** Some objects cannot be associated with an advanced filter, and the option will be disabled when that is the case for an object selected in the dialog.

Checking the option **Group By Object Type** will consolidate all the items by categories. All subtypes will automatically be included as a consequence. The filters are still available.

If the box **Only Show Objects in Display or Already Selected** is checked, only the items or categories (depending on whether the Group By Object Type option is checked) of objects existing or selected in the respective oneline diagram will be shown in the supplied list.

The buttons **Check All** and **Uncheck All** are useful to select/unselect all the items/groups in the list. The button **Check Only Text Fields** is available only when the Group by Object Type is unchecked, and when clicked on all the items that have text associated to them are selected, and the rest of the items are unselected. Clicking on the **Reset To Defaults** button will remove all the current selections and reset them to the default settings.

If you wish to select objects that *do not* have the chosen criteria, then check **Select All Except What Meets the Above Criteria**.

If you wish to only select objects that are currently visible on the diagram (due to layering, etc.), check the box labeled **Select only currently visible objects** to apply the criteria settings only to objects on the diagram that are currently visible.

If the Select by Criteria Dialog is open and some display objects have already been selected, an advanced selection option becomes available. If the box labeled **Use as a filter on presently selected objects** is checked, the criteria chosen in the Select by Criteria Dialog *will only* affect the previously selected objects. In other words, only the objects that were previously selected AND that

match the chosen criteria will remain selected when the OK button is pressed. For example, you could use all the options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects on the oneline. Then using the Select by Criteria Dialog, choose the Bus criteria and check **Use as a filter on presently selected objects** to select only the buses from the rectangular selected group of objects.

Once you have selected the criteria and pressed *OK*, all objects that satisfy ALL the criteria will be chosen. If the Select All Except What Meets the Above Criteria is unchecked, then the selected objects will lie within the specified areas AND zones, the specified voltage level, and the specified zoom levels, and will be of one of the drawing types specified.

You can save your settings for your Select by Criteria session with the case by clicking **Save As** to save with a new name or **Save** to save with the current name. This allows you to re-open the Select by Criteria dialog, and quickly recover settings you may have previously used. Use the **Rename** to rename an already saved set, or **Delete** to remove a previously saved set of criteria.

If you have saved different criteria sets, you can export them all to an [Auxiliary file](#) by choosing **Save to AUX file**. The **Load All from AUX file** will load an entire auxiliary file, regardless of whether it contains Select by Criteria settings or not.

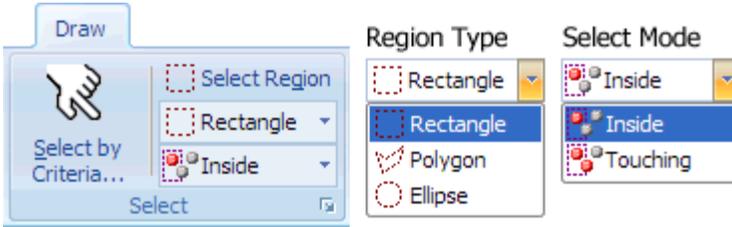
If you wish to clear all settings you have modified on the dialog, click the **Reset to Defaults** button.

Using this dialog in conjunction with the [Format Selection Dialog](#), can be a very fast and easy way to customize your displays.

NOTE: When choosing to select all tie lines, all objects associated with the tie line will also be selected. This includes all line fields for the line as well as the two buses that are the endpoints of the line.

Select Ribbon Group

The Select ribbon group contains options to help select multiple online objects simultaneously. The features in this topic are only available when you are in [Edit Mode](#). This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



The features available on this ribbon group are described below.



Select by Criteria

Click on the Select by Criteria button to open the [Select by Criteria Dialog](#). On online diagrams of even a modest size, this is an extremely useful feature and PowerWorld highly encourages you to use this feature.

Select Region

Use to select all objects in a particular region of the oneline. When the **Region Type** is **Rectangle** or **Ellipse**, then after clicking the button, click and hold the left mouse button on the oneline at the point where you would like to begin the selection. Then, drag the mouse to size the selection shape. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. When the Region Type is Polygon, then after clicking this button single click on your oneline diagram for each vertex of the polygon. Finally, finish the selection by double-clicking the mouse after which all objects inside your polygon will be selected.

Depending on the **Select Mode** below, every object located entirely **Inside** the region, or if you have chosen **Touching**, then every object inside or partially touched will be selected. Selected objects will have handles appear indicating that they have been selected.

Region Type

Specify a **Rectangle**, **Polygon**, or **Ellipse**. This controls the shape used to select objects on the oneline diagram when using the **Select Region** button above.

Select Mode

This combo box specifies how the **Select Region** button above works. Choose **Inside** in order to select only the objects completely inside the selection rectangle. Choose **Touching** in order to select every object either partially touched by or completely inside the selection rectangle.

There are also other ways to select multiple objects

- You can also select multiple objects manually by holding down the SHIFT key which clicking on objects in Edit Mode. This will also add to your selection.
- If you hold down the SHIFT and CTRL keys simultaneously and then click on your oneline diagram and select a region it has the same affect as click on the **Select Region** button and selecting a region on your oneline

Grid/Highlight Unlinked Objects

The Grid/Highlight Unlinked Objects tab is only available in Edit mode. It controls the appearance of a grid in the oneline. A grid is not visible by default, but can be setup from the Grid/Highlight Unlinked Objects page of the [Oneline Display Options](#) dialog. Additional options are available from this dialog for disabling anchors and highlighting unlinked objects on oneline diagrams.

Snap Objects to Grid

If checked, new objects placed on the oneline will be aligned with the grid; the grid does not need to be visible on the oneline to align objects to it.

X Grid Spacing, Y Grid Spacing

These determine the horizontal and vertical spacing of the grid.

Display Grid Lines on Oneline

If checked, the grid lines will appear on the oneline

Horizontal Show Every, Vertical Show Every

These numbers determine the density of the visible gridlines; for fewer lines enter higher numbers.

Gridline Color

Click in the rectangle to choose the color of the gridlines. The default is grey.

Anchor Options

A check box is available for temporarily disabling anchors on a oneline diagram. This option will disable the anchor properties of all objects on the oneline diagram.

Do not prompt regarding relinking objects after dragging

Check this check-box to avoid being prompted whether to relink a graphical object after this has been dragged.

Highlighting of Unlinked Objects

Highlight Unlinked Objects with Color

Checking this box allows for any objects on all open oneline diagrams that are currently not linked to any data in the case to be highlighted using the highlight color. To change the highlight color used, left-click in the color box to choose a different color.

Minimum Highlighted Object Pixel Size

The minimum size, in pixels, of the highlight image. This is to prevent the highlight from being unnoticeable when the zoom level is very low.

Extra Width for Highlighted Lines (pixels)

Sets an extra width parameter, in pixels, for highlighting line objects. This will make line object highlights appear wider than other highlighted objects.

Display Unlinked Elements in Run Mode

This option allows the highlighting of unlinked elements when you switch to run mode. By default the highlighting of unlinked elements is restricted to edit mode, to avoid confusion in run mode, particularly when drawing [contours](#).

Using the Oneline Alignment Grid

The oneline alignment grid allows for buses and other objects to be easily aligned when placed onto a oneline diagram. The alignment grid for a oneline diagram can be configured in the Oneline Display Options dialog under the [Grid/Highlight Unlinked](#) page. You can open this display by selecting **Oneline Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab.

When the alignment grid is enabled most objects will snap to the grid while being moved on the oneline. Depending on whether snap to grid is enabled or disabled, objects can individually be made to do the *opposite* of the snap to grid setting by holding down the Alt key while dragging the object.

Setting Background Color

The Background Color Dialog changes the background color for the display. To view this dialog, select **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab, or right-click in the background of the oneline and select **Online Display Options** from the local menu. Switch to the [Display Options](#) tab, click **Change Background Color**, or click on the colored rectangle to bring up the Color Dialog, which you can then use to select the new background color. Click OK to register the new color. The Online Display Options dialog will provide a preview of the color you selected. To define this color as the default background color to use in all new onelines you create, click the **Set as Default Background Color**. Finally, click OK to save your color selection.

Zoom, Pan, and Find

The Zoom, Pan, and Find Options Dialog is used to specify a desired zoom level and screen location or to locate a particular bus or area/zone on the oneline. The dialog can be opened by right-clicking on the background of the diagram and choosing **Pan/Zoom Control** from the oneline popup menu, or by clicking on the **Find** button in the [Zoom](#) ribbon group on the [Onelines](#) ribbon tab. The dialog has the following fields:

Find Objects on Oneline

The Find Object on Oneline section of the dialog is used to pan to any desired [bus object](#), [area/zone object](#), [branch object](#), [interface object](#), or [substation object](#) on the oneline. The dialog makes use of the advanced find functionality of Simulator to locate a device.

Only Include Objects Visible at Current Zoom Level

If checked (the default), then the **Object ID** combo box only lists those objects that are visible at the current zoom level.

Object Type

Select the type of object to find: *Buses, Areas or Zones, Interfaces, Substations and Lines/Transformers.*

Sort By

Specify whether you are entering the object by its number or by its name by choosing either **Sort by Name** or **Sort by Number**. This option also determines how the entries in the list of objects are sorted (either by number or by name).

Object ID

Enter the object's number or name (depending on the sort type chosen) or select the object from the list.

Allow Auto Updating on Selection

Check this box to automatically pan to the specified object. The object will be located at the center of the oneline diagram.

Pan to Object on Oneline

Click this button to pan to the specified object if the Auto Updating box is not checked. The object will be located at the center of the oneline diagram.

Auto-Zoom when Panning

Check this box to automatically change the zoom when panning to the specified object.

Zoom / Pan

The Oneline Zoom / Pan tab on this dialog is used to allow the user to specify either a new zoom level and/or screen center and to define these as the new display default settings. The zoom level and screen center can also be changed from the keyboard. Please see [Online Zooming and Panning](#) for details.

Zoom Level

Enter the desired percentage zoom level (nominal is 100%).

Horizontal, Vertical

Enter a desired location for the center of the screen. The nominal screen center is 50 horizontal and 50 vertical.

Pan/Zoom to New Location

Changes the screen center and zoom level to the values specified in the above fields.

Restore Default Values

Resets the zoom level and screen center to the default values.

Default Drawing Values

The [Default Drawing Options Dialog](#) allows you to see and change the various default values used to create new objects. To show this dialog select **Default Drawing Values** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab.

Formatting Ribbon Group

The **Formatting** ribbon group contains options for formatting the selected objects on your active oneline. This group is on the [Draw](#) ribbon tab and its options are available only in Edit mode. The options allow you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. The group is shown below.



A description of the actions on this group are as follows.



Format Buttons

Clicking this button will open the [Format Dialog](#). The dialog which opens will have five tabs as follows.

Font

Select this option to change the font with which the selected objects are displayed. The [Font Tab](#) of the Format Selection Dialog is displayed.

Line/Fill

Select this option to change the line thickness and style with which a display object is drawn, or the fill color for closed shapes. The [Line/Fill Tab](#) of the Format Selection Dialog is displayed.

Levels/Layers

Select this option to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize. The [Levels/Layers Tab](#) of the Format Selection Dialog is displayed.

Display/Size

Simulator can change the attributes of the way multiple objects are displayed using this option, which opens the [Display/Size page](#) of the Format Selection Dialog. The size of the objects can be adjusted, as well as the orientation of the object(s) from their terminal buses. Objects can also be Anchored to their terminal buses or devices, or marked as Immobile, meaning the object(s) cannot be moved on the oneline diagram.

Field

Select this option to change the attributes related to object text fields which are selected. The [Field Tab](#) of the Format Selection Dialog is displayed.

Anchors

Invokes a command to [refresh anchors](#) on all oneline objects.



Copy Format and Paste Format Buttons

The button on the left is the **Copy Format** button. After selecting an object this will copy all the format properties of that object into an internal buffer. When you then subsequently select other objects, you can click on the button on the right which is the **Paste Format** button. Clicking the **Paste Format** button will bring up the [Paste Format Dialog](#).



Send to Back, Bring to Front Buttons

After selecting a group of online objects using any of the methods described on the [Select](#) ribbon group, you may then click the **Send to Back** button to cause all the objects selected to move to underneath other objects. Click the **Bring to Front** button to bring objects to the top.

The Send to Back and Bring to Front menu options govern the visibility of display objects that occupy the same screen stack level. All objects have an associated screen stack level, which may be one of Base, Background, Middle, or Top. Objects having a screen stack level of Top obscure all objects having stack levels of Middle, Background, or Base that occupy the same region of the screen. They may or may not obscure objects having the same stack level of Top depending on the order in which the objects having the same stack level are drawn. Selecting Send to Back for a selected object will cause it to be obscured by all other same-level objects that occupy its location. Selecting Bring to Front for a selected object will cause it to be drawn above all other same-level objects that occupy the same region of the screen. The Send to Back and Bring to Front menu options govern relative placement of objects only *within* stack levels. The Send to Back and Bring to Front options do not affect the relative placement of objects having different stack levels.

Alignment Menu

The alignment menu provides access to the [Alignment Dialog](#) and direct access to options to align the selected objects as specified in the menu items.

Grouping Menu

Grouping is only available for background online objects (background lines, ellipses, text, etc...). This menu provides button to Group and Un-Group objects.

Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active online diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



Format Multiple Objects

The Format Selection Dialog features five tabs of controls for modifying the display attributes of selected objects. To open the Format Multiple Options dialog you must first use some of the options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects. Next, right click on any of the selected objects and select **Format Selection** from the local menu. You may also bring up the Format Multiple Objects dialog by going to the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

Line/Fill

The [Line/Fill Tab](#) is used to change the line size, color, and style and the fill color with which the selected objects are drawn.

Levels/Layers

The [Levels/Layers Tab](#) is used to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize.

Display/Size

The [Display/Size Tab](#) controls the size and orientation of the selected objects.

Font

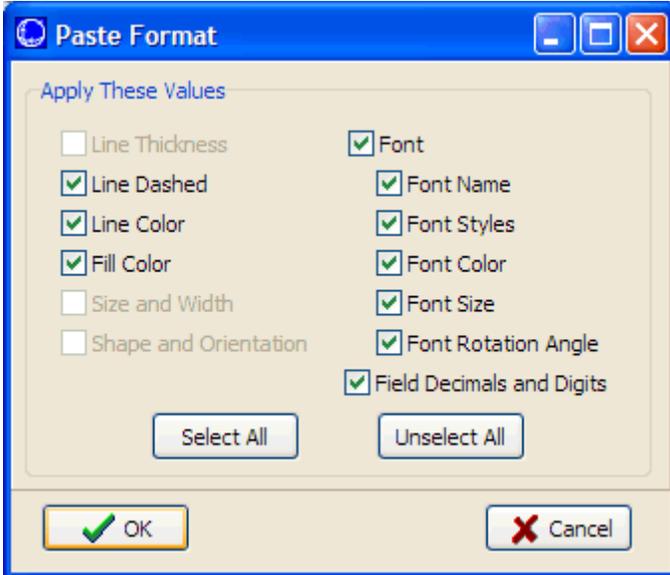
The [Font Tab](#) is used to change the font used in text/font objects.

Field

The [Field Tab](#) is used to change selected fields on the diagram to a different field designation. If all of the selected fields are for the same type of object and field, you can select a different field to be represented by the selected objects from the **Field** selection. You can also set the options for including a suffix, the total digits in the field for numeric values, and the number of digits to the right of the decimal.

Paste Format Dialog

The Paste Format Dialog appears when you use the Paste Format button on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. The dialog looks as follows



On this dialog check the attributes you would like to paste to the selected objects. Note that attributes which are disabled in the Paste Format Dialog represent one of two things

- The attribute was not relevant to the objects whose format was copied
- The attribute was not the same for the objects whose format was copied

Font Properties

The Font Tab allows you to define the font for the selected objects by choosing Font type, size, color and effects. The page also allows you to control the default display font using the **Make Default** button.

By clicking on **Make Default**, the current font name, size, and effects will be set as the default font for the display. This has the same effect as opening the [Default Drawing Values](#) and setting the default font.

Line/Fill Properties

The Line/Fill Tab is used to customize the line size/color and the fill color of selected objects. You can view this tab by selecting  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This tab's controls can be used to change the appearance of most, but not all, screen objects. When you modify the line thickness or color of an object, these new values will be set as the default settings for all new line objects. You can view the case defaults using the [Default Drawing Options](#) Dialog. The Line and Fill Options tab has the following fields/commands:

Line Thickness

Thickness of the line in pixels.

Dashed

Allows setting a line to appear as a dashed line on the diagram. There are three types of line dashing to choose from in the drop down menu for this option.

Note: Due to drawing limitations, the line thickness for a dashed line **must be equal to 1**. Any other line thickness will result in the line being drawn as solid, regardless of the dashed setting.

Line Color

The Line Color field shows the current line color. To change the line's color, click on the box displaying the current line color, or click the **Change Line Color** button. This displays the Color Dialog. Select the desired color and select OK to change the color, or click Cancel if you do not wish to change the color.

Line Color 2

The Line Color 2 field applies only to transformer objects. Transformers can be represented by different colors on each side of the transformer coils. This is commonly used to color match each side of the transformer with the color used to represent the voltage level of the transmission system on each particular side of the transformer. However, you can customize the colors for selected transformers by modifying them here.

Note that the Line Color 2 field ALWAYS applies to the high voltage side of the transformer when you change

Use Background Fill

Click on this field to toggle whether or not to fill the background for the selected objects with the selected background fill color. Only text objects such as [Text](#), [Bus Fields](#), [Generator Fields](#), [Load Fields](#), [Switched Shunt Fields](#), [Line Fields](#), [Transformer Fields](#), and [Area Fields](#) and background objects such as [Background Lines](#) and [Background Rectangles](#) can be filled.

Fill Color, Change Fill Color

The Fill Color field shows the current fill color. To change the fill color, click on the box that displays the current fill color, or click the **Change** button. This displays the Color Dialog. Click on the desired color and the select OK to change the color, or click Cancel if you do not wish to change the color.

Levels/Layers Options

The Levels/Layers Tab is used to customize the stack level or layer an object is contained in. To set layers for objects, select the objects on the oneline diagram (edit mode only) and then choose . The Levels/Layers Options page of the [Format Multiple Objects](#) dialog will be displayed, with the following settings available.

Stack Level

An object's stack level dictates what objects it will appear above, and which objects it will appear below on a oneline diagram. For example, circuit breaker and pie chart objects have a default stack level of Top. Therefore anything with a stack level of Middle, Background or Base will appear underneath pie charts and circuit breakers on the oneline diagram. Objects that are within the same stack level and are drawn in the same location will result in the last object drawn being the visible object on the diagram. You can toggle which elements within the same stack level at the same location is visible using the Send to Back and Bring to Front buttons in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

Layers

Layers are different than stack levels in that layers are designed to allow the user to filter the elements visible on a diagram (in run mode). Initially all diagram objects are in the same layer, called Default. This default layer cannot be modified or deleted. However, you can introduce additional layers using either the **Add New...** or **Define...** buttons. Clicking on **Add New...** will open a dialog to name the new layer, and will automatically add the new layer to the case and change the Layer name in the drop down list to the new layer. Clicking on **Define...** will open the [Screen Layers](#) list display, which will allow you to manage the full list of layers defined for the diagram. Once layers have been defined, you can choose which layer the selected objects belong to by choosing the Layer name from the drop-down list by clicking on the down arrow to the right of the layer name.

Settings for resizing when zooming

You can modify the maximum and minimum zoom levels at which the selected text fields will no longer resize. Thus when zooming in or out on the diagram, text fields will resize according to the zoom level until the minimum or maximum zoom level are reached. At that point, text fields will no longer resize, but will stay fixed at their current size as you continue to zoom.

Maintain Fixed Screen Location (do not pan)

This option is available for text fields. When checked for selected text fields, these text fields will no longer pan when you pan the rest of the diagram. This allows you to place text that will always be visible, regardless of what part of the diagram you are observing. You can still move the individual text field itself in edit mode.

Maintain Fixed Screen Size (do not resize on zoom)

This option is available for text fields. When checked for selected text fields, these text fields will no longer resize when the zoom level of the diagram is changed. This allows you to place text that will always be visible at a constant size, regardless of what zoom level you are currently observing on the oneline diagram. You can still select the text fields themselves in edit mode and change their font size.

Screen Layers

The Screen Layers display can be invoked from either the [Levels/Layers](#) button in the [Formatting](#) ribbon group in Edit Mode, or by right-clicking on the oneline diagram background and choosing Edit Screen Layers.

Screen layers provide a method to filter objects on a oneline diagram based on either zoom level or to which layer the objects are assigned. Initially each object is in a common layer called Default, which cannot be modified or deleted. You can create new layers using the Screen Layers display.

Since the Screen Layers display is a [Case Information Display](#), it shares the of other case information displays. You can access many case information display features by right-clicking on the display to invoke the local popup menu. Most importantly, this menu contains the options to Insert and Delete screen layers.

To delete a screen layer, right-click on that layer in the display and select Delete from the popup menu. The one exception is the Default layer, which cannot be deleted.

To insert a new screen layer, right-click on a record in the display and choose Insert from the popup menu. This will open the [Screen Layer Options](#) dialog for defining the options of the new layer you are inserting. You can also define a new layer by going to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **Screen Layer > Define Screen Layer**.

Screen Layer Options

The Screen Layer Options are used to define options for new layers, or modify options for existing layers. Devices can be added to a layer by choosing the layer name from the drop-down list on the [Levels/Layers Tab](#) of the [Format Multiple Objects](#) dialog.

Name

Layers can have any name desired to describe the layer. The default naming convention is "Layer #".

Show Layer

If this box is checked, any objects on the diagram which are contained in this layer *may* be visible. This depends on the settings for low and high zoom level, described below. If the box is unchecked, however, then any objects contained in this layer will be hidden. Only objects that are visible will be used when calculating contours. Unlike [Custom Display Detail](#), layers can be applied in both edit mode and run mode.

Selectable in Edit Mode

By unchecking this box the user prevents the objects contained in this layer of being with the mouse during Edit Mode. This behavior is particularly desirable for elements (such as borders which are pretty much already fixed) when editing online diagrams.

Use Conditional Display by Zoom Level

Objects can be displayed or hidden based on the Screen Layer to which they are assigned and the current zoom level of the online diagram. Checking this box, and setting the Low Zoom Level and High Zoom Level fields, dictates at what zoom range the objects contained in the zoom level will be visible. If the zoom level is outside the range defined, any objects belonging to the screen layer will be hidden. This also applies to both edit mode and run mode.

Memo

This tab allows the user to enter text in order to describe this layer.

Format Field Properties

The Field tab of the [Format Multiple Objects](#) dialog can be used to modify the format of the selected fields, or to change a set of selected fields from one field representation to another. Use the [Select by Criteria](#) dialog to select fields of a particular type, right-click on any of the selected fields and choose **Format Selection** from the popup menu.

Note that this option also applies to objects which display information related to specific object fields, such as [Line](#) and [Interface](#) Pie Charts.

Object Type

This field will be filled in automatically based on the type of object the selected field(s) apply to. You cannot change this field, which means that you can only change the selected field(s) to another field for the same type of object.

Field

The field type of the selected field(s). To change the selected fields to a new field type, click on the drop-down arrow of the combo box, or click on the **Find...** button, and select a new field from the list of fields for the Object Type.

Include Suffix

Checking this option will add the unit suffix to the selected fields.

Total Digits in Field

Modify the number of digits to display in the field. This number includes the decimal and the numbers to the right of the decimal.

Digits to Right of Decimal

Modify the number of the total digits that should appear to the right of the decimal.

Display/Size Properties

Use the Display/Size Tab of the [Format Multiple Objects](#) dialog to change the display size, orientation, anchor property of the selected objects.

Size

This field will change the characteristic display size of the selected objects. The characteristic size is normally the vertical size or height of display objects. In objects with different orientation choices, the size is the most distinctive dimension of the object (e.g. in buses with right or left orientations, the size is the horizontal dimension of the bus bar, but in buses with up or down orientations, the size is the vertical dimension of the bus bar).

Width

This field will change the characteristic display width of the selected objects. Similarly to the size, the width represents the horizontal dimension or width of display objects. In objects with different orientation choices, the width is the least distinctive dimension of the object (e.g. in buses with right or left orientations, the width is the vertical dimension of the bus bar, but in buses with up or down orientations, the width is the horizontal dimension of the bus bar).

Anchored

To toggle whether or not the selected objects are [anchored](#), check or uncheck this option

Immobile

To force the selected objects to stay in the same position, check this option.

Orientation

This group will modify the orientation setting of the selected objects.

Shape

Use this group to modify the shape of the selected display objects. If the option selected is a valid selection for an object, the object shape will be changed; otherwise, the original shape will not be modified. For Substation objects, the Use Substation Layout Settings specifies that the shape selected in the [Substation Display Options](#) will be used.

Number of Selected Objects

This field shows the number of objects that are currently selected on the oneline.

Align Group Objects

The Align Group feature allows for selecting multiple objects on a online diagram and aligning them in a specified manner. There are a couple of ways to perform the alignment.

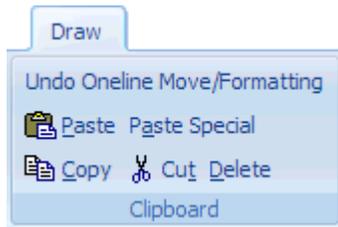
First, select the objects on the diagram that you wish to align. Once you have selected all objects, you can 1) right click on a selected object and choose **Align Group Objects** from the local popup menu or 2) use the **Alignment** menu found in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

The **Align Group Objects** option from the local menu will open the Align Dialog, which allows for specifying the type of alignment desired. Alignment can be performed horizontally, vertically, or both. In addition to alignment of the top, bottom, left and right edges of the selected objects, you can also choose to have the objects snap to the [drawing grid](#) in the horizontal and/or vertical directions as well.

These alignment actions can also be selected directly from the **Alignment** menu found on the Formatting ribbon group on the Draw ribbon tab.

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting online objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



Undo

The Undo command is used in the edit mode to undo the last change made on the online diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the online. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

Absolute Coordinates : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different online diagrams which contain the same geographic background.

Coordinates Relative to Cursor : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

Undo Command

The Undo command is used in the edit mode to undo the last change made on the online diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Chapter 7: Properties of Simulator Objects

This chapter provides a description of the properties of data and display objects. Of the many properties of the various objects in Simulator, some are relevant only to the Edit Mode, and some only to the Run Mode.

Note that when you are using the Simulator, you can obtain context sensitive help for oneline objects by positioning the cursor on the desired object and pressing the F1 key. Context sensitive help can also be obtained by pressing the F1 key over many of the displays and dialogs as well.

The following material is included:

- Edit Mode Properties and Information
- Run Mode Properties and Information
- General Properties and Information

Bus Information (Edit Mode)

This dialog is used to view/modify information about each bus in the system during Edit Mode. It is very similar in content to its [Run Mode](#) counterpart.

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus. You can use the spin button immediately to the right of the number to move to the next bus (click the up arrow) or the previous bus (click the down arrow).

If adding a new bus to the one-line display, this dialog will open after selecting the location of the bus display object. This dialog can be used to add a bus to the one-line for buses that already exist in the power flow case or new buses that should be created. If entering the bus number of a bus that does not already exist in the case, an informational message will appear above the Bus Number field indicating that using this bus number will insert a new bus into the power system data model and not just to the one-line display.

Find By Number

To find a bus by its number, enter the number into the **Bus Number** field and then click this button.

Bus Name

Unique alphabetic identifier for the bus.

Find By Name

To find a bus by its name, enter the bus name into the Bus Name field (case insensitive) and then click this button.

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Nominal Voltage

Nominal voltage of the bus in kV.

Labels

Clicking on this button will open the [Label Manager Dialog](#) listing all of the labels assigned for the selected bus.

Area Number

Number of the bus' area. Each bus must be associated with an area record. If a bus is specified as belonging to an area that does not already exist in the model, the new area is created.

Area Name

Alphabetic identifier for the bus' area. If the area already exists, you do not need to enter this value.

Zone Number

Number of the bus' zone, between 1 and 2,147,483,647 (equals 2^{31} minus 1). Each bus must be associated with a zone record. If a bus is specified as belonging to a zone that does not already exist in the model, the new zone is created. Zones provide a useful mechanism for breaking up a large system. Buses can be assigned to zones independent of their area assignments. Thus, a single area could contain multiple zones, or a single zone could span multiple areas. You can use the [Zone](#)

[Dialog](#) to list the buses in a particular zone and easily move a group of buses from one zone to another.

Zone Name

Alphabetic identifier for the bus' zone. If the zone already exists, you do not need to enter this value.

Owner Number

The number of the bus' owner.

Owner Name

The name of the bus' owner.

Substation Number

The number of the substation the bus is contained in. This is typically blank in most power flow data, unless specifically added to a PowerWorld binary save file.

Substation Name

The name of the substation the bus is contained in. This is typically blank in most power flow data, unless specifically added to a PowerWorld binary save file.

OK, Save, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the **Find By Number** command to edit additional buses. **Cancel** closes the dialog without saving any changes.

Bus Information

Voltage (pu), Angle (degrees)

Current per-unit voltage magnitude and angle for the bus. If you are inserting a new bus into an existing system, **you should not change the initial per unit voltage values**. Rather, when you first switch to Run Mode, Simulator will estimate the voltage magnitude and angle at the new buses in such a way as to reduce the initial mismatches. **This automatic estimation is only available if you have not modified the voltage in any way.**

System Slack Bus

Check only if the bus should be modeled as a system slack bus. Each case requires at least one slack bus. Simulator can also dynamically determine slack buses as described in the [Advanced Power Flow Solution Options](#).

Display

Orientation

Set the orientation of the bus. The current choices are right, left, up or down. For the current bus object shapes of Rectangle and Ellipse, right and left are analogous to horizontal, up and down are analogous to vertical. Additional shapes for buses may be available in the future for which right, left, up and down may have more specific impact on the appearance of the bus object.

Shape

Sets the shape of the bus object.

Width

Specifies the horizontal axis of an elliptical bus or horizontal side length of a rectangular bus.

Size

Specifies the vertical axis of an elliptical bus or the vertical side length of a rectangular bus.

Scale Width with Size

When this option is checked, changing the size will cause the width to automatically adjust to keep the same ratio of width to size. To adjust the width independent of the size, uncheck this option or adjust the width separately after adjusting the size.

Link to New Bus

If you have right clicked on a bus and opened the bus information dialog, you could change the bus number in the **Bus Number** field and press this button to force the bus object on the diagram to link to the new bus number and information in the load flow data.

Attached Devices

Load Summary Information

Displays the net MW and Mvar load at the bus. You cannot change either of these fields from this display. Select the Add or Edit Bus Load Records to view the individual load records for the bus via the [Load Dialog](#).

Shunt Admittance

The real (G) and reactive (B) components of shunt compensation at the bus, expressed in MW and MVR, respectively.

Fault Parameters

This tab is only visible when viewing the bus information for a bus with attached load. The parameters on this tab are used when running a fault analysis study. The values represent the total load at the bus for the negative and zero sequence as equivalent admittances. By default, these values are zero. For load buses, these values can be changed by the user, or they can be specified by loading short circuit data from within the [Fault Analysis Dialog](#).

Custom

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Bus Field Information

Bus field objects are used primarily to indicate various quantities associated with bus devices. Furthermore, some bus field types, which are distinguished by an integrated spin button, may be used to change bus device properties.

The Bus Fields Information Dialog can be used to modify the properties of individual bus fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus associated with the field. Use the drop-down box to view a list of all buses in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus associated with the field. Use the drop-down box to view a list of all buses in the case with [valid area/zone/owner filters](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

This option is currently not used for bus fields.

Maintain Constant Load Power Factor

This option is only used for certain fields, such as load fields and generator fields. It is not used currently for any bus fields.

Include Suffix

If this checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Anchored

If this checkbox is checked, the bus field is [anchored](#) to its associated bus, which means that it will move with the bus.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Type of Field

Used to determine the type of bus field to show. The following choices are available:

Bus Name Name of the bus

Bus Number Bus number (from 1 to 2,147,483,647 (equals 2^{31} minus 1))

Bus Voltage Bus voltage magnitude

Bus Angle Bus voltage angle in degrees

MW Marginal Cost Bus MW marginal cost in $\$/\text{MW}\cdot\text{hr}$; available only with OPF

Mvar Marginal Cost Bus Mvar marginal cost in $\$/\text{Mvar}\cdot\text{hr}$; available only with OPF

MW Loss Sensitivity Increase in losses for 1 MW of generation over the nominal

Select a Field Choose from any defined Bus fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

Shortest Path Between Elements

To access the Shortest Path Between Elements go to the [Tools](#) ribbon tab, and choose **Connections > Determine Shortest Path Between...** from the [Other Tools](#) ribbon group. The Determine Shortest Path Between dialog provides a way to find the shortest electrical pathway between two parts of the power system.

After clicking the **Calculate** button, the shortest path of AC branches will be determined with the first bus being inside the Start Element and the last bus being inside the End Element. Options described below allow the user to choose the Start Element, End Element, Distance Measure for each branch, and branches that are allowed to be traversed. The results will be displayed in a case information display at the bottom of the dialog showing the sequence of buses that are traversed between the start and end elements.

The options on the dialog are described below.

Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

End Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the ending element. For more help on the object choose see the [Find Dialog Basics](#).

Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X - Per unit series reactance.

|Z| - Magnitude of the series impedance (based on the per unit reactance and resistance).

Length - Length field for each branch.

Number of Nodes - Length of 1.0 is used for all branches.

Other - When choosing Other, click the **Find..** button to choose any numeric field of a branch.

Lines to Process

Regardless of the Distance Measure above, you can choose which branches are allowed to be traversed when finding the shortest path.

All - All branches are allowed to be traversed.

Only Closed - Only branches that are presently closed can be traversed.

Filter - Only branches that meet the advanced filter specified can be traversed. Click **Define...** to choose or create an advanced filter.

Selected - Only branches whose Selected? field is set to *YES* can be traversed.

Substation Information (Edit Mode)

The Substation Dialog is used in the [Edit Mode](#) to view information about a substation and to move one or more buses from one substation to another. (See [Substation Information \(Run Mode\)](#) for help on the corresponding [Run Mode](#) version.) To view this dialog, first select **Aggregation > Substations** from the [Model Explorer](#) to view the [Substations Records](#) Display. Then, right-click on the desired substation record and select **Show Dialog** to view this dialog.

The dialog has the following fields:

Substation Number

An integer identifier for the substation. You can use the spin button immediately to the right of this field to move to either the next substation (click the up arrow) or the previous substation (click the down arrow).

Substation Name and ID

Two alphanumeric identifiers for the substation. You can use these fields to change the substation's name or ID, provided you click either Save or OK.

Find By Number

To find a substation by its number, enter the number into the **Substation Number** field, then click this button.

Find By Name

To find a substation by its name, enter the name into the **Substation Name** field, then click this button.

Find By Sub ID

To find a substation by its substation ID, enter the ID into the **Substation ID** field, then click this button.

Find...

If the exact substation number, name and ID are not known, you can use the [Find Dialog](#) to search for and select a substation from a list of substations.

Labels

Clicking this button will open the [Subscribed Aliases dialog](#) displaying the list of defined labels for the substation. New labels can also be added for the substation from the dialog as well.

Buses

This table lists all of the buses in the substation. Number, name, voltage, area number and name, and zone number and name are shown for each bus. This table can be used to move buses to a different substation. Select the bus or buses you would like to move with the mouse. Then, enter the Destination Substation Number, which is the substation to which you want to move the selected buses. You may enter a substation number that does not already exist, too, so that the buses will be moved to a brand new substation. In this case, be sure to provide the new substation a name and ID, as well. Finally, click the Move Selected Bus(s) to Destination Substation button to implement the move.

Display Options

The Display Options tab of the Substation Information dialog is only visible if an open diagram contains an object representing the substation. This tab allows you to choose the general appearance

of the substation object. Use the Width and Height fields to set the **width** and **height** of the substation object. The **Shape** field allows you to choose what shape the substation object will take.

By clicking on **Link to New Object**, the Choose Object selector will appear, allowing the user to select another substation in the load flow data to be linked to this particular graphical object.

The **Substation Layout Online, URL or Command** field allows you to specify the same of the Simulator online diagram (pwd) file that should be automatically opened if you click on the substation object on the diagram in run mode. If no online is specified, Simulator will search for a default name of the format AreaName_SubstationName. If a diagram cannot be found, then Simulator will not attempt to open any online diagrams when the substation is clicked. Optionally, a HTTP URL can be entered in the form <http://example.com>. Also, a line command can be entered, such as *Excel.exe c:\file.xls*.

Substation Generators, Loads, and Switched Shunts Tables

These tables list all of the generators, loads, and switched shunts in the substation. Number, name, ID, status, and additional fields for each type of device are shown.

Custom

Enter any text notes you wish in the Memo page. When the case is saved as a Simulator PWB file, the memo text will also be saved. Custom fields can also be entered for storage with the selected substation.

Geography

Displays geographic information about the location of the substation, in Latitude and Longitude. When showing the dialog by right-clicking on a [substation online display object](#) from a online diagram, then a button will also appear that states **Copy Longitude/Latitude from Online Location**. This button will then determine the longitude/latitude of the center of the substation object on the online diagram and copy these values into the Substation Data Record.

OK, Save, Cancel

OK saves any changes to the substation name or ID, and closes the dialog.

Save saves any changes to the substation name or ID, but does not close the dialog.

Cancel closes the dialog ignoring any changes.

Substation Field Options

Substation field objects are used to show different values associated with substations. This dialog is used to view and modify the parameters associated with these fields.

Substation Number

Select the number of the substation for which you are inserting or viewing information of a substation field.

Find...

If you do not know the exact substation you are looking for, you can click this button to open the [advanced search engine](#).

Substation Name

The name of the currently selected substation.

Substation ID

The substation ID number.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Anchored

Toggle the anchoring of the field to the substation object of which it is associated.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of super area field to show. The following choices are available:

Substation Name, Substation Number

Name or number of the selected substation.

Substation ID

ID string for the selected substation.

Max Nominal Voltage

Displays the nominal voltage of the highest nominal voltage bus in the substation.

Substation Load MW, Substation Load Mvar

Total Load MW or MVAR in the substation.

Substation Gen MW, Substation Gen MVAR

Total Generator MW or MVAR in the substation.

Select a Field

Select this option and click **Find Field...** to display the value of any other substation specific field.

Generator Information (Edit Mode)

This dialog is used to view and modify the parameters associated with each generator in the system. It can also be used to insert new generators and sometimes to delete existing generators.

The Edit Mode version of the Generator Information Dialog is almost identical to the [Run Mode](#) version.

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the generator is attached. The dropdown list enumerates all generator buses in the case that meet the criteria established by [area/zone/owner filters](#). You may select a bus number directly from the dropdown list, or you may use the spin buttons to cycle through the list of generator buses.

Bus Name

Unique alphabetic identifier for the bus to which the generator is attached, consisting of up to eight characters. Use this dropdown box to view a list of all generator bus names in the case with valid [area/zone/owner filters](#).

ID

Two-character alphanumeric ID used to distinguish multiple generators at a bus; '1' by default.

Find By Number

To find a generator by its number and ID, enter the number into the **Bus Number** field and the ID into the **ID** field. Then click this button.

Find By Name

To find a bus by its name and ID, enter the bus name into the **Bus Name** field (case insensitive) and the ID into the **ID** field. Then click this button.

Find...

If you do not know the exact generator bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the generator, either Closed (connected to terminal bus) or Open (not connected). You can use this field to change the status of the generator.

Area Name

Alphabetic identifier for the terminal bus' area.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected generator.

Fuel Type

Type of fuel used by the generator this model represents. In most cases, this field is unnecessary for normal load flow analysis, and hence the default value is Unknown. However, this value can be useful during the [Security Constrained OPF](#) analysis.

Unit Type

The type of unit the generator represents, such as combined cycle, steam, hydro, etc.

There are six additional areas of information on this dialog for specific aspects of generation:

[Display Information](#)

[Power and Voltage Control](#)

[Costs](#)

[Fault Parameters](#)

[Owner, Area, Zone, Sub](#)

[Custom](#)

Generator Field Information

Generator field objects are used primarily to indicate various quantities associated with generation devices. Furthermore, some generator field types, which are distinguished by an integrated spin button, may be used to change generation device properties.

The Generator Fields Information Dialog can be used to modify the properties of individual generator fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus to which the generator associated with the field is connected. Use the dropdown box to view a list of all buses with generators in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus to which the generator associated with the field is connected. Use the dropdown box to view a list of all buses with generators in the case with [valid area/zone/owner filters](#).

ID

ID of the generator associated with the field. Generator ID are two-character alphanumeric fields.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

Generator fields can be used not only to show various fields associated with generation devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the generator field. When the up spin button is clicked, the generator field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the generator field value is decreased by the same amount.

This field is only used for fields of the following types: Setpoint Gen MW, Setpoint Gen Mvar, and Gen Setpoint Voltage. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the generator field on the oneline.

Rotation Angle in Degrees

The rotation angle at which the text field should be displayed.

Anchored

If the *Anchored* checkbox is checked, the generator field is [anchored](#) to its associated generator, which means that it will move with the generator.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of generator field to show. The following choices are available:

Gen MW Output MW generation

Gen Mvar Output Mvar generation

Gen AGC Status AGC status of generator; status can be toggled in Simulator

Gen AVR Status AVR status of generator; status can be toggled in Simulator

Gen Setpoint Voltage Voltage of generator on bus

Select a Field Choose from any of the different generator fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

Generator Options: Display

This information is located on the [Generator Information Dialog](#).

Display Size

The size of the generator.

Scale Width with Size

If checked, the Display Width will automatically be scaled to the appropriate setting when the Display Size is changed. If unchecked, then only the length of the generator object will be affected by changing the value of the Display Size.

Display Width

The width of the display object. This setting is automatically set if Scale Width with Size is checked and the value of the Display Size field is changed, or the Display Width value can be set manually to a new value.

Pixel Thickness

Thickness of the display object in pixels.

Orientation

Specifies the direction in which to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

Link to New Generator

Links the object to a different generator in the data.

Generator Options: Power and Voltage Control

This information is located on the [Generator Information Dialog](#).

Power Control

The Power Control grouping fields are used to show/change the values associated with the real power output of the generator.

MW Output

Current real power output of the generator.

Minimum and Maximum MW Output

Minimum and maximum real power output limits for the generator. Simulator will not let the MW output go below its minimum value or above its maximum value if the *Enforce MW Limits* option is exercised.

Available for AGC

Determines whether or not the generator is available for automatic generation control (AGC). Normally this box should be checked. However, there are times when you would like to control the generator output manually (such as if you are using the generator to remove a line limit violation), in which case you should leave this box unchecked. A generator is also placed on "manual" control anytime you manually change its output. You could then place the generator back on AGC control if you wish it to participate in an area generation dispatch.

Enforce MW Limits

If checked, the minimum and maximum MW limits are enforced for the generator, provided the **Enforce Generator MW Limits** field is also checked on the Limits Tab of the [Simulator Options Dialog](#). If this box is checked and a generator is violating a real power limit, the generator's MW output is immediately changed.

Participation Factor

The participation factor is used to determine how the real power output of the generator changes in response to demand when the generator is available for AGC and the area is on [participation factor control](#). When you open a case using the [PTI Raw Data Format](#), this field is initialized to the per unit MVA rating of the generator, since participation factor information is not stored in the PTI format.

Voltage Control

The Voltage Control grouping is used to show/change values associated with controlling the voltage/reactive power output of the generator.

Mvar Output

Current reactive power output of the generator. You can manually change this value only if **Available for AVR** is not checked.

Min and Max Mvar Output

Specify the minimum and maximum allowable reactive power output of the generator.

Available for AVR

Designates whether or not the generator is available for automatic voltage regulation (AVR). When the AVR field is checked, the generator will automatically change its reactive power output to maintain the desired terminal voltage within the specified reactive power range. If a reactive limit is reached, the generator will no longer be able to maintain its voltage at the setpoint value, and its reactive power will then be held constant at the limit value.

Use Capability Curve

If checked, then the generator's reactive power limits are specified using a reactive capability curve that prescribes the dependence of the generator's reactive power limits on its real power output. Otherwise, the fixed values given in the **Min Mvar Output** and **Max Mvar Output** fields are used. The generator reactive capability can be defined using the table that appears at the bottom of the dialog. Please see [Generator Reactive Power Capability Curve](#) for details.

Regulated Bus Number

Number of the bus whose voltage the generator is regulating. This is usually, but not always, the generator's terminal bus. Multiple generators can regulate the same remote bus, but the regulated bus must not be another generator bus. If the generator is at a slack bus, it must regulate its own terminal voltage. Select **Solution Details > Remotely Regulated Buses** in the [Model Explorer](#) to view the [Remotely Regulated Buses](#) display, which identifies all buses that are being remotely regulated.

SetPoint Voltage

Specifies the desired per unit setpoint voltage value the generator is to regulate at the regulated bus. The regulated bus need not be the generator terminal bus.

Remote Reg %

This field is only used when a number of generators at different buses are regulating a remote bus (i.e., not their terminal buses). This field then specifies the percentage of the total reactive power required by the remote bus to maintain its voltage that should be supplied by this generator. The default value is 100. If the total value is different from 100%, then all the regulation factors are normalized to obtain a percentage of regulation.

Wind Control Mode and Power Factor

The Wind Control Mode effects how generator Mvar limits are treated and can be set to either *None*, *Boundary Power Factor* or *Constant Power Factor*.

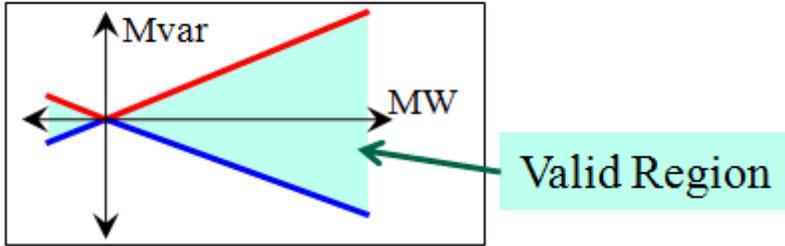
When the mode is *None*, then the generator behaves using the standard settings of the **Min and Max Mvar Output** and **Capability Curve** described above.

For both the *Boundary* and *Constant* modes, the Mvar limit magnitudes are determined from the actual **MW Output** and the **Wind Control Mode Power Factor** value.

$$\text{Magnitude} = \text{MWOutput} * \tan(\arccos(\text{Power Factor}))$$

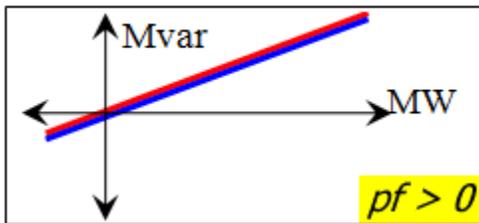
For *Boundary* mode, the **Max Mvar Output** is positive and the **Min Mvar Output** is negative. This provides a boundary under which the Mvar must operate.

Boundary Power Factor

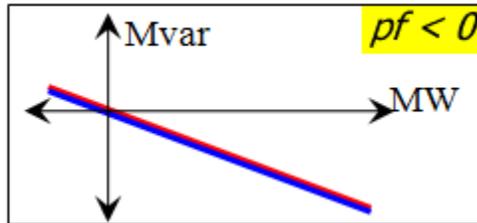


For *Constant* mode, **Max Mvar Output** and the **Min Mvar Output** are made the same with a positive **Wind Control Mode Power Factor** meaning the limits have the same sign as the actual **MW Output**, and a negative **Wind Control Mode Power Factor** meaning the limits are the opposite sign as the actual **MW Output**, thus the **Mvar output** is a function of **MW Output** and **Power Factor**.

Constant Power Factor



Constant Power Factor



Generator Options: Costs

The **Costs** tab of the Generator Information dialog (run mode) is used to show/change values associated with the cost of operation of the generator. See [Generator Cost Information](#) for details. Cost data can also be saved/loaded using the [Generator Cost Data files](#).

Output Cost Model

Cost Model

Simulator can model generators as not having a cost model, or having either a cubic cost model or a piecewise linear model. The cost model type you choose determines the content of the remainder of this dialog

Unit Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

Variable O&M

The Operations and Maintenance costs. Only used for cubic cost models.

Fixed Costs (costs at zero MW Output)

The fixed costs associated with operating the unit. These costs are independent of the generator's MW output level and is added to the cost prescribed by the cubic or piecewise linear model to obtain the total cost of operating the generator in \$/MWhr. The total fixed cost is the addition of the fuel cost independent value and the fuel cost dependent value multiplied by unit fuel cost.

Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form $C(P_{gi}) = (d * P_{gi}^3 + c * P_{gi}^2 + b * P_{gi}) * (\text{fuel cost}) + \text{fixed costs}$ (as described above), specify the cost curve's coefficients. The A coefficient is historically the fuel cost dependent fixed cost value, which is combined separately now with the fuel cost independent value. These coefficients can be specified only when you have chosen to use a cubic cost model.

Piecewise Linear Table

If you have chosen to use a piecewise linear cost model, a table appears that allows you to specify pairs of MW output levels and corresponding generator operating costs. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

Convert Cubic Cost to Linear

Use this option to create a piecewise linear cost function from the cubic cost function specified by the coefficients A, B, C, and D and the fuel cost. Specify the number of break points, and hence the number of segments, in the **Number of Break Points** field. Click the **Convert to Linear Cost** button to create the piecewise linear function that approximates the cubic cost function. This action switches **Cost Model** option to *Piecewise Linear* and displays the **Piecewise Linear Table** that identifies the piecewise linear curve's breakpoints.

Bid Scale / Shift

Cost Shift, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Marginal Cost (run mode only)

Shows the marginal cost of producing real power at the generator at its current output level, $dC_i(P_{gi})/dP_{gi}$.

ED/OPF Cost (run mode only)

This is the cost of production for this generator following an economic dispatch or optimal power flow solution, *including* the scaling from the cost shift and cost multiplier fields.

Unscaled Cost (run mode only)

The cost of production of the generator, *ignoring* the cost multiplier and cost shift. This cost is the result of the original cost function by itself.

OPF Reserve Bids

This tab is only available with the [OPF Reserves add-on](#). Detailed information about this tab can be found in the [Generator and Load OPF Reserves Bids](#) topic.

Generator Options: Fault Parameters

The parameters on this tab are used when running a [fault analysis](#) study.

Generator MVA Base

The assumed MVA base for the generator. This value is used when calculating fault analysis values for the internal generator parameters.

Neutral Grounded

Check this check-box if the generator has the neutral grounded.

Generator Step Transformer

The resistance, reactance and tap setting for the generator step-up transformer, if one is being modeled internally with the generator. By default, no internal transformer model is assumed.

Internal Impedance

These fields represent the internal impedance of the generator for all three sequences. By default, all three values are initially the same as the load flow internal impedance of the generator. All three sets of values can be modified, either manually or by loading values from an external file using the [Fault Analysis Dialog](#).

Neutral-to-Ground Impedance

Neutral-to-ground impedance for the generator. These values get implemented with the zero sequence admittance matrix. Note that the neutral-to-ground impedance will not be used, even if specified, if the original model for the generator implicitly models the generator step-up transformer. This is because the implicitly modeled transformer is assumed to have a delta winding on the generator side of the transformer, which isolates the generator from the rest of the zero sequence network.

Generator Options: Owners, Area, Zone, Sub

This information is located on the [Generator Information Dialog](#).

This tab is used to display or change the generator's owner information, area information, zone information, and substation information.

Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal buses' owner.

Owners

Currently, Simulator supports up to four owners for generators. To add an owner of a generator, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. To modify an owner's percentage of ownership, simply modify the value in the percentage field for that owner. If you set the percentage of an owner to 0, that owner will be removed from the list of owners for the device. You can also remove an owner from owning part of a device by changing the owner field for that owner to 0. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the generator dialog.

Area Number, Area Name

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The name and number of the substation to which the generator belongs, and it is the same than that of the terminal bus.

Generator Cost Description

The cost associated with operating a generator varies according to the output of the generator, with the general rule that getting more power out of a generator costs more. In Simulator, there are two options for modeling generator cost. The first employs the common cubic relationship

$$C_i(P_{gi}) = \text{IndFixedCost} + (a_i + b_i P_{gi} + c_i (P_{gi})^2 + d_i (P_{gi})^3) * \text{fuel cost } \$/\text{Hour}$$

where P_{gi} is the output of the generator at bus i in MW. IndFixedCost is the fuel cost independent fixed cost, and therefore is not multiplied by the fuel cost. The value a_i is the fuel cost dependent fixed cost, and is therefore multiplied by the fuel cost. The values b_i , c_i , and d_i are used to model the generator's input-output (I/O) curve. The I/O curve specifies the relationship between how much heat must be input to the generator (expressed in MBtu per hour) and its resulting MW output. Normally, the cubic coefficients remain constant for a generator. The last term in the equation is the fuel cost, expressed in $\$/\text{MBtu}$. This value varies depending on the fuel used in a generator. Typical values would be $\$ 1.25/\text{MBtu}$ for coal and $\$ 2/\text{Mbtu}$ for natural gas. The resultant equation is known as the fuel-cost curve. The value of the fixed costs, b_i , c_i , d_i , and fuel cost can be viewed and modified using the [Generator Information Dialog](#).

Simulator can also model generator costs using a piecewise linear model consisting of pairs of MW output and incremental cost ($\$/\text{MWhr}$) of generation, along with a fixed cost. These piecewise linear curves must be convex curves, meaning the marginal cost of the current MW break point must be higher than the previous MW break point.

Such curves can be defined using the [Generator Information Dialog](#) or by loading data from [generator cost data](#) files.

Set Generator Participation Factors

Participation factor control is another of Simulator's mechanisms for distributing an area's responsibility to serve its load, losses, and interchange. It is particularly well-suited to implementing [automatic generation control \(AGC\)](#) when you do not have good economic information for an area's generators. With participation factor control, the amount of power that each generator contributes to meeting its areas load, loss, and interchange responsibilities is controlled by the size of its participation factor. The unit that has the largest participation factor contributes the most, and the unit that has the smallest participation factor contributes the least.

The **Set Generator Participation Factors Dialog** gives you a convenient way to define the participation factors for multiple generators. You can set the participation factor according to a number of different formulae and then apply this prescription to all generators in a specific area, all generators in a specific zone, all generators in the system, or all generators whose display filters are currently set to true.

To display the **Set Generator Participation Factors Dialog**, you first need to open the [Area Information Dialog](#) and switch to the Area MW Control Options page. The Area Information Dialog has a button labeled **Set Participation Factors** that is enabled only if the Participation Factor Control option is selected under the Area Control Options heading. Set the area on participation factor control by selecting the Participation Factors option, and then press the **Set Participation Factors** option.

The Set Generator Participation Factors Dialog is divided into two parts. The first part, which occupies the top half of the form, allows you to indicate how the participation factors should be calculated or set for each generator. Your options include:

Max MW Rating of Generator The participation factor for each generator is set to the generator's maximum MW capability.

Difference Between Max and Current Output The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute.

Constant Value of The participation factor for each generator is set to the same hard-coded value.

File The participation factor for each generator is read from a file. The first line of the file should contain the keyword NUMBERS or NAMES indicating whether generators are identified by bus number or by bus name in the file. All subsequent lines should be comma-delimited and contain three fields: the number or name of the generator's bus, the generator's id, and the generator's participation factor.

If you choose any of the first three options, you then must tell Simulator to what generators you want to assign the participation factors. To assign the participation factors to all generators in a specific area, select the **All Generators in Area** option, and then choose the area from the adjacent dropdown box. If you want to assign the participation factors to all generators in a specific zone, select the **All Generators in Zone** option, and then choose the zone from the adjacent dropdown box. If you want to assign the participation factor to all generators in the system regardless of their area or zone affiliation, select the **All Generators in System** option. Finally, if you want to assign the participation factor to just those generators whose display filter criteria evaluates to true, choose the **All Generators With Valid Display Filters** option.

If you instead chose to read participation factors from a file, only those generators whose factors you read from the file will have their factors set by this action. However, unless each generator's associated area is set to control generator output using participation factor control, this information will be ignored. To make sure that each generator's area is set to participation factor control, check

the **Set Corresponding Areas to Participation Factor Control** box. Then, each corresponding area will be set to participation factor control.

Generator Reactive Power Capability Curve

The reactive power output of most generators depends on the real power output of the generator. This dependence is expressed using a reactive capability curve. Simulator models the reactive capability curve using a piecewise linear approximation. The reactive capability curve is modified on the [generator dialog](#) and can be saved/loaded using the [Generator Reactive Capability Curve Auxiliary files](#).

Modeling a Reactive Power Capability Curve From the Generator Dialog

- Make sure the **Use Capability Curve** checkbox is checked.
- In the table immediately below the checkbox, prescribe the reactive capability curve using up to 50 points. The points should be ordered by MW, in numerically increasing order. At each point specify the MW value, the minimum reactive power value in Mvar, and the maximum reactive power value in Mvar. The first point should correspond to the minimum MW output of the generator, while the last point should correspond to the maximum MW output.
- To insert a new point, click on the desired column, and then right-click to display the table's local menu. Select Insert Point.
- To remove a point, click on the desired column, and then right-click to display the table's local menu. Select Delete Point.
- When finished be sure to select **Save** to save your modifications.

In the Run Mode, you can view the reactive power capability curve graphically by right-clicking on the generator to display its submenu and then selecting **Reactive Capability Curve**.

Load Information (Edit Mode)

This dialog is used to view and modify the parameters associated with each load in the system. It can also be used to insert new loads and sometimes to delete existing loads. It is nearly identical in structure to its [Run Mode](#) counterpart.

The Load Information Dialog can be used to inspect and modify the model of a bus load. To view the Load Information Dialog, select the load and choose **Show Dialog** from the [case information toolbar](#) or right-click on the load of interest and select **Load Information Dialog** from the resulting local menu. The dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the load is attached. The dropdown box provides a list of all load buses with valid [area/zone/owner filters](#). You can use the spin button to cycle through the list of load buses.

When you insert objects graphically, the Bus Number and Bus Name fields are usually set automatically to the bus upon which you placed the object.

Bus Name

Unique alphabetic identifier for the bus to which the load is attached, consisting of up to eight characters. The dropdown box lists the names of all load buses in the case with valid [area/zone/owner filters](#).

ID

Two-character ID used to distinguish multiple loads at a bus. By default, the load ID is equal to "1 ." An identifier of '99' is used to indicate an equivalent load.

Find By Number

To find a load by its number and ID, enter the number into the Bus Number field and the ID into the ID field. Then click this button.

Find By Name

To find a load by its name and ID, enter the bus name into the Bus Name field (case insensitive) and the ID into the ID field. Then click this button.

Find...

If you do not know the exact load bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the load, either Closed (connected to terminal bus) or Open (not connected). You can use this status field to change the load's status.

Labels

Clicking on this button will open the [Label Manager](#) dialog listing all the labels assigned for the selected load.

Area Number, Area Name

Number and name of the area the load is a member of.

Zone Number, Zone Name

Number and name of the zone the load is a member of.

Substation Number, Substation Name

Number and name of the substation the load and the terminal bus are members of.

Owner Number, Owner Name

Number and name of the owner the load is a member of. If the load owner is the same as the terminal bus owner, the **Same Owner as Terminal Bus** box will be checked. Loads DO NOT have to be owned by the same owner as the terminal bus.

OK, Save, Delete, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the Find By Number command to edit additional loads. **Delete** deletes the current load. **Cancel** closes the dialog without saving your changes.

Specific load information can be found on the following pages of the Load Options dialog:

[Load Information](#)

[OPF Load Dispatch](#)

[Custom](#)

Load Options: Load Information

This page of the Load Options dialog contains information on the load magnitude and display settings.

MW and Mvar Value Fields

The MW and Mvar Value fields are used to represent the amount of base real and reactive load at the bus. Usually this load is modeled as being "constant power," meaning that the amount of load is independent of the bus voltage magnitude. However, Simulator also permits modeling "constant current" load, for which the load varies in proportion to the bus voltage magnitude, and "constant impedance" load, for which the load varies in proportion to the square of the bus voltage magnitude. Values in these fields are specified in MW and MVR assuming one per unit voltage. All six fields can be modified by the user.

Display Size

Size of the load.

Scale Width with Size

If checked, the Display Width will automatically be scaled to the appropriate setting when the Display Size is changed. If unchecked, then only the length of the generator object will be affected by changing the value of the Display Size.

Display Width

The width of the display object. This setting is automatically set if Scale Width with Size is checked and the value of the Display Size field is changed, or the Display Width value can be set manually to a new value.

Pixel Thickness

Thickness of the display object in pixels.

Orientation

Specifies the direction to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

Link to New Load

Links the object to a different load record in the data.

Load Options: OPF Load Dispatch

This tab of the Load Options dialog contains settings for allowing the load to be included as an OPF Control. The load(s) can then be dispatched in the OPF algorithm according to the assigned costs.

Benefit Model**Benefit Model**

If this field is set to none, the load will not be dispatchable in the OPF solution. If the option is set to Piecewise Linear, the load is dispatchable during the OPF, according to the following fields.

Min. and Max. MW Output

Minimum and maximum load MW demand for OPF dispatch.

Available for AGC

If checked, the load will be available for redispatch during the OPF routine.

Fixed Benefit

Value of the load benefit at minimum demand.

Piece-wise Linear Benefit Curve

This table allows you to specify pairs of MW demand levels and corresponding load benefit values, which in turn define the starting points and slopes of the piece-wise linear benefit curve segments. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

OPF Reserve Bids

This tab is only available with the [OPF Reserves add-on](#). Detailed information about this tab can be found in the [Generator and Load OPF Reserves Bids](#) topic.

Load Field Information

Load field objects are used primarily to indicate various quantities associated with load devices. Furthermore, some load field types, which are distinguished by an integrated spin button, may be used to change load device properties.

The Load Fields Information Dialog can be used to modify the properties of individual load fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus to which the load associated with the field is connected. Use the dropdown box to view a list of all buses with loads in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus to which the load associated with the field is connected. Use the dropdown box to view a list of all buses with loads in the case with [valid area/zone/owner filters](#).

ID

ID of the load associated with the field. Load ID fields are two characters in length.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

Load fields can be used not only to show various fields associated with load devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the load field. When the up spin button is clicked, the load field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the load field value is decreased by the same amount.

This field is only used for fields of the following types: Load MW and Load Mvar. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the load field on the oneline.

Note that the **Maintain Constant Load Power Factor** option will allow you to specify a Delta per Mouse-click for the MW load, and when the MW value is changed in run mode, the MVAR load will also change in such a way as to keep the power factor of the load constant.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Anchored

If the *Anchored* checkbox is checked, the load field is [anchored](#) to its associated load, which means that it will move with the load.

Rotation Angle in Degrees

The angle at which the text is placed on the diagram, in degrees.

Type of Field

Used to determine the type of load field to show. The following choices are available:

Load MW MW load

Load Mvar Mvar load

Select a Field Choose from any of the different load fields

Select **OK** to save changes and to close the dialog, or click Cancel to close the dialog without saving your changes.

Load Modeling

Each load can be modeled as having voltage variation. The voltage variation is modeled using the Base Load Model fields on the [Load Information](#) dialog. Thus the actual real and reactive value of each load is determined using the following equation:

$$MW = \text{Load MW Multiplier} * (\text{SMW} + \text{IMW} * V + \text{ZMW} * V * V)$$

$$\text{Mvar} = \text{Load Mvar Multiplier} * (\text{SMvar} + \text{IMvar} * V + \text{ZMvar} * V * V)$$

where

MW current real power load in MW

Mvar current reactive power load in Mvar

SMW constant power MW value

Smvar constant power Mvar value

IMW constant current MW value (assuming 1.0 per unit voltage)

Imvar constant current Mvar value (assuming 1.0 per unit voltage)

ZMW constant impedance MW value (assuming 1.0 per unit voltage)

Zmvar constant impedance Mvar value (assuming 1.0 per unit voltage)

V per unit bus voltage magnitude

Load MW Multiplier = (Area MW Multiplier) * (Zone MW Multiplier)

Load Mvar Multiplier = (Area Mvar Multiplier) * (Zone Mvar Multiplier)

The Area Multipliers scale all of the loads in an area. The Zone Multipliers scale all of the loads in a zone. These are static values that do not vary over time. If a time varying schedule of loads is required, the [Time Step Simulation](#) tool should be used instead of the static multipliers.

Line/Transformer Information (Edit Mode)

The Line/Transformer Information Dialog is used to view and modify the parameters associated with each transmission line and transformer in the system. You can also insert new transmission lines and delete existing transmission lines from this dialog.

The Edit Mode version of this dialog is very similar in content to its [Run Mode](#) counterpart.

The Line/Transformer Information Dialog has the following fields:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side.

To Bus Number and Name

To Bus number and name.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Find By Numbers

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

Find By Names

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

Find...

If you do not know the exact from and to bus numbers or names you are looking for, you can click this button to open the [advanced search engine](#).

From End Metered

This field is only used for lines and transformers that serve as tie lines, which are lines that join two areas. If this field is checked for a tie line, then the *from* end of the device is designated as the metered end. Otherwise the *to* end is metered. By default, the *from* end is metered. The location of the metered end is important in dealing with energy transactions because it determines which party must account for transmission losses.

Default Owner (Same as From Bus)

Read-only check-box that indicates whether the line's owner is the same than the from bus' owner.

From and To Bus Area Name

Control area name in which each terminal bus is located.

From and To Bus Nominal kV

Nominal voltage level of each terminal bus, in kV.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected branch.

OK, Save, Delete, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the *Find By Number* command to edit additional transmission lines. **Delete** deletes the current transmission line. **Cancel** closes the dialog but does not save any changes.

Display

See [Transmission Line/Transformer Display Options](#).

Parameters

See [Transmission Line/Transformer Parameters Options](#).

Transformer Control

This tab is only visible for transformer objects. See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

Series Capacitor

This tab is only visible for series capacitor objects. See [Series Capacitor](#) for details on modeling series capacitors.

Fault Info

The parameters on this tab are used when running a [fault analysis](#) study. The values represent the zero sequence impedance and zero sequence line shunt admittances for the analysis. By default, the positive and negative sequence line impedances and line shunt admittances are the same as the load flow impedance. The same fields are used for transformers, along with the configuration field. The configuration field defines the winding type combinations for the transformer (wye, delta, etc.) As a default, Simulator assumes a grounded wye to grounded wye transformer, which has the same model as a transmission line. Usually transformers are not of this type, and the proper type would need to be defined either manually or loaded from an external file in order for the fault analysis to be accurate.

Owner, Area, Zone, Sub

The **Default Owner (Same as From Bus)** read-only check-box indicates whether the line's owner is the same than the from bus' owner. Transmission elements can have up to four different owners, each with a certain owner percentage. To add an owner of a transmission element, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the line/transformer dialog.

The area, zone and substation to which the From and To buses belong, are also shown.

Custom

This page of the dialog can be used to enter notes about the transmission line. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be populated with user-specified values and will be saved with the case as well.

Transmission Line/Transformer Options: Display

Pixel Thickness

Thickness of the display object in pixels.

Anchored

If checked, the object is anchored to its terminal bus, which means that it will move when you move the terminal bus. See [Anchored Objects](#) for details.

Link to New Line

Use the *Link to New Line* button to create a new line corresponding to the entries you have made in the dialog. This button performs the same function as pressing *Save*. Note that adding a new line to the case in this way does not add a transmission line to the oneline display; the new line is present only in the model. You may then add the newly modeled line to the oneline diagram in the usual way (such as select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab).

Symbol Segment

Only visible for transformer and series capacitor objects. This field specifies which "segment" of the branch contains the transformer or capacitor symbol. A segment constitutes a section of the line between vertex points of the line object, and are numbered starting at the from bus.

Symbol Size

Only visible for transformer and series capacitor objects. Specifies the size (width) of the transformer or capacitor symbol on the branch.

Symbol Percent Length

The distance or "length" of the symbol on the segment of the branch which contains it.

Show Detailed Line Vertices

When checking this option, a new group of information appears, showing the x,y coordinates of every vertex of the line. Additionally to being able to edit those locations, it is possible to shift or scale all the values with the buttons **Shift All Values** and **Scale All Values**, respectively.

Transmission Line/Transformer Options: Parameters

Status

Current status of the device.

Length

If the length of the line is known, it can be entered here for informational purposes.

Per Unit Impedance Parameters

The series resistance and reactance, the total charging susceptance (that is, B , not $B/2$) and the shunt conductance of the device (in per unit). In the case of transformers, the magnetizing conductance and susceptance can also be specified.

Line Shunts

Select to view the [Line Shunt](#) Dialog. This dialog is used to only view or modify the values of the line shunts in Run Mode. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer. If the line has shunts, the check box **Has Line Shunts** is checked.

MVA Limits

Ratings for the transmission line or transformer in MVA. Simulator allows the use of up to eight different limits.

Calculate Impedances

This will display a pop-up menu with the following two options:

...From Per Distance Impedances

Clicking this item will open the [Line Per Unit Impedance Calculator dialog](#), which can be used to convert actual impedance and current limits to per unit impedance and MVA limits, and vice versa.

...From Conductor Type and Tower Configuration

Clicking this item will open the [Transmission Line Parameter Calculator](#) dialog, which can be used to compute per unit impedance values, based on a conductor type and a tower configuration.

Convert Line to Transformer

Clicking this button turns the currently selected transmission line into a transformer, making the transformer specific fields available.

Line Shunts Information

The Line Shunts Information Dialog is used to modify the parameters of transmission line shunts. The modeling of the line shunts is explained below.

Line Shunts

Line shunts are included in the model as admittance-to-ground values at either the From end or the To end of a transmission line. These values can represent many things, such as shunt-to-ground capacitors, reactors, zigzag (or grounding) transformers, or equivalenced system values. Line shunt values are entered in per unit, with a positive B corresponding to capacitors and a negative B corresponding to reactors. Mathematically, the line shunts are included in the algorithm in the same manner that line charging capacitance is included using the Pi model.

Line shunt values can be modified from the [Transmission Line Information](#) dialog.

Line Per Unit Impedance Calculator Dialog

The Line Per Unit Impedance Calculator Dialog allows to convert actual impedance and current limit values to per unit impedance and MVA limit values, and vice versa.

The dialog has the following elements:

Actual Impedance and Current Limits

This part of the dialog shows all the impedance related values in Ohms/length-unit as well as the transmission line limits specified in Amps. If any of these values is modified, the corresponding per unit or MVA value will be changed accordingly, taking into consideration the current line length, length units, and system base values.

Line Length

This value indicates the length of the line in miles or kilometers, depending on the units selected in **Length Units**. The option **When changing convert** is used to convert the values when the length value is modified. The option **PU/MVA -->** indicates that the actual impedance and current limits will be converted to per unit impedance and MVA limits when the length value is changes. The option **<-- Electrical** specifies that the per unit impedance and MVA limits will be converted to actual impedance and current limits when the length value is modified.

Length Units

This option indicates the length units. Choices are miles and kilometers. When this parameter is changed, the user will be prompted to confirm to convert the actual impedance and current limits, as well as the line length values, from the old units to the new units. If the answer is positive, then the actual impedance and current limits, and the line length will be the same but they will be expressed in the new units selected. If the answer is negative, the values will not change numerically but they still will be expressed in the new units selected.

System Base Values

The system base values show the power base, the voltage base, and the impedance base. These values can not be modified in this dialog.

Per Unit Impedance and MVA Limits

This part of the dialog shows all the impedance related values in per unit as well as the transmission line limits specified in MVA. If any of these values is modified, the corresponding actual or Amps value will be changed accordingly, taking into consideration the current line length, length units, and system base values.

Conductor Type

Shows the conductor type selected to compute the per unit impedances in the [Transmission Line Parameter Calculator](#) dialog. It will be blank if there isn't any.

Tower Configuration

Shows the tower configuration selected to compute the per unit impedances in the [Transmission Line Parameter Calculator](#) dialog. It will be blank if there isn't any.

Calculate PU Impedances From Conductor Type and Tower Configuration

Clicking this button will open the [Transmission Line Parameter Calculator](#) form.

Transmission Line Parameter Calculator

The transmission line parameter calculator is a tool designated to compute characteristic line parameters give the type of the conductor and the configuration of a three-phase overhead transmission line.

The parameters computed are the resistance R , reactance X , susceptance B , and conductance G . These values are computed as distributed (per unit of distance), lumped or total (for a specific line distance), and in per-unit.

Calculations

The following controls are part of the calculations section:

Parameters Calculation

This section is to enter the necessary data to compute the characteristic line parameters that are shown in the Results panel.

Input Data

Conductor Type: This is the combo box that lists all the conductor types available in the Conductors table. To add, remove or edit a specific conductor and its characteristics, see Conductor Type section below.

Tower Configuration: This combo box lists all the tower configurations available in the Tower Configurations table. To add, remove or edit a specific tower configuration, please go to the Tower Configuration section below.

Line Length: This is the value of the distance of the transmission line. The units are miles when using English system, or kilometers when using the Metric (SI) system.

Line Length Units: The line length units specify the measurement system to use when entering the line length. The options are English system or Metric (SI) system. The final and intermediate results will also be shown in the system specified here.

Power Base: The system voltampere base in MVA.

Voltage Base: The line-line voltage base in KV.

Impedance Base: The impedance base in Ohms. This value is automatically computed when the power base and the voltage base are entered or modified.

Admittance Base: The admittance base in Siemens. It is also automatically computed as the inverse of the impedance base.

Final Results

When all the input data is entered, the results automatically will be displayed. The values for R , X , B and G are shown in three different sections, each section corresponding to Distributed, Lumped or Total, and Per Unit values, respectively. The Power Surge Impedance Loading is calculated only for the lumped section.

Intermediate Results

The intermediate values calculated to compute the R , X , B , and G values are displayed here. The computed geometric mean radius and geometric mean distance are shown in the Distributed

values section. The characteristic impedance and propagation factor are displayed in the Lumped values section.

Note: To see the specific calculations used in this program, see the Calculations section, at the end of this document.

Ampere to MVA Limit Conversion

This section converts the limits of the transmission line from Amperes to MVAs, given the voltage base, and vice versa.

Conductor Type

This section is used to add, remove, rename, and edit the information related to the conductor types. This can be done in two ways: using the form for an individual conductor type, or using the table for all the conductor types available.

Edit By Form

Conductors are identified by a unique code word. All the available conductors are listed in the Conductor Code Word combo box. By selecting one conductor, all its properties are displayed in the form. There, the user is able to modify any of those values. After modification of any value, the user has to save the changes by clicking on the button **Save** before changing tabs, otherwise the changes will be lost.

By clicking on **New**, a message prompting for a name for a new conductor will be shown. By clicking on **Rename**, a new name for the current conductor type is required. In order to save the current conductor type with a different name is necessary to click on **Save As**. Finally, to remove the current conductor the user can do so by clicking on the **Delete** button.

Edit By Table

In this tab, all the conductor types are shown as records in a table, where every field is a characteristic of the conductor. In order to edit the records in this table, use the Database button described in the Database section. While editing the table, the user can not change of tab until changes are posted or discarded.

Conductor Properties

The available properties of the conductors are the following:

Code Word: Code name for the type of conductor. The names of bird species are typically used. Code Word has to be unique.

Area: The area of aluminum conductor in circular mils. A circular mil is the cross-sectional area of a circular conductor having a diameter of 1 mil. One mil is one thousandth of an inch (0.001").

Aluminum strands: Number of aluminum strands.

Steel layers: Number of steel strands.

Aluminum layers: Number of aluminum layers.

External diameter: Outside diameter of the conductor in inches.

GMR: Geometric Mean Radius in feet.

DC Resistance: DC resistance of the conductor at 20°C per 1 mile in Ohms.

AC Resistance 25: AC resistance of the conductor at 60 Hz and 25°C per 1 mile in Ohms.

AC Resistance 50: AC resistance of the conductor at 60 Hz and 50°C per 1 mile in Ohms.

AC Resistance 75: AC resistance of the conductor at 60 Hz and 75°C per 1 mile in Ohms.

Inductive Reactance: Inductive reactance per conductor at 1 foot spacing at 60 Hz in Ohms/mile.

Capacitive Reactance: Capacitive reactance per conductor at 1 foot spacing at 60 Hz in MegaOhms/mile.

Tower Configuration

This part is used to add, remove, rename, and edit the information related to the tower configurations. This can be done in two ways: using the form for an individual tower configuration, or using the table for all the tower configurations available.

Edit By Form

Tower configurations are identified by a unique name. All the available tower configurations are listed in the Tower Configuration Name combo box. By selecting one specific tower configuration, all its characteristics are displayed in the form. There, the user can modify any of those characteristics. After modification of any value, the user has to save the changes by clicking on the button **Save** before changing tabs, otherwise the changes will be lost.

By clicking on **New**, a message prompting for a name for a new tower configuration will be shown. By clicking on **Rename**, a new name for the current tower configuration is required. In order to save the current tower configuration with a different name is necessary to click on **Save As**. Finally, to remove the current tower configuration the user can do so by clicking on the **Delete** button.

Edit By Table

In this tab, all the tower configurations are shown as records in a table, where every field is a value of the tower configuration. In order to edit the records in this table, use the Database button described in the Database section. While editing the table, the user can not change of tab until changes are posted or discarded.

Tower Configuration Values

The values of the tower configuration are the following:

Name: Name for the tower configuration. Name has to be unique.

Phase spacing: x and y coordinates of phases A, B and C positions. Values are in feet for English system and meters for Metric (SI) system. When these values are modified, the phase spacing graph is automatically updated. **Draw axis** has to be checked for x and y axis to be drawn in the graph.

Conductors per bundle: This section specifies the number of conductors (either single conductor or a bundle of conductors) per phase. The maximum number of conductors per bundle allowed is 4.

Use regular spacing: When using a bundle of conductors, checking the **Use Regular Spacing of** check box will use the specified value as a regular spacing among the conductors. If the **Use Regular Spacing of** check box is not checked, then the custom conductors spacing section has to be used.

Conductors spacing: x and y coordinates of the conductors in the bundle. Values are in feet for English system and meters for Metric (SI) system. When these values are modified, the bundle spacing graph is automatically updated. **Draw axis** has to be checked for x and y axis to be drawn in the graph.

Temperature: Assumed temperature in Fahrenheit degrees for English system and Celsius degrees for Metric (SI) system.

Frequency: Frequency of the system in Hertz.

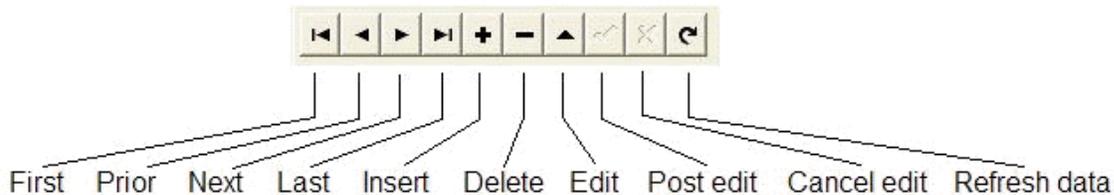
System of units: The system of units used to specify the values of the tower configuration. The options are English system or Metric (SI) system. The units specified here not necessarily have to math the units specified in the **Input Data** section.

Database

The conductor type and tower configurations tables are read by default from the file **conductors.mbd**, which is a MS Access® database. This database can be read from another *.mdb file by clicking on the **Select Conductors and Configurations Database** button.

Note: The conductors.mdb file can also be viewed and edited in MS Access®.

In order to edit a record in the database tables, the user can use the toolbar designed to do that. Following there is a figure showing this toolbar:



The **First**, **Prior**, **Next**, and **Last** buttons are used to move among records. The **Insert**, **Delete** and **Edit** buttons are used to insert, delete or edit the current record, respectively. While editing a record, the user can not change of tab until modifications are posted through the **Post edit** button or canceled with the **Cancel edit** button. The **Refresh data** button just refreshed the data of the entire table.

Calculations

Distributed Parameters

Resistance

$$R_t = \frac{\left(R_{25} + \left(\frac{R_{25} - R_{50}}{25 - 50} \right) (t - 25) \right)}{N} \quad 0 \leq t \leq 50$$

$$R_t = \frac{\left(R_{50} + \left(\frac{R_{50} - R_{75}}{50 - 75} \right) (t - 50) \right)}{N} \quad 50 \leq t$$

Where:

R_t AC resistance at temperature t per phase per 1 mile in Ohms

t Assumed temperature in Celsius degrees

R_{25} AC resistance of the conductor at 60 Hz and 25°C per 1 mile in Ohms

R_{50} AC resistance of the conductor at 60 Hz and 50°C per 1 mile in Ohms

R_{75} AC resistance of the conductor at 60 Hz and 75°C per 1 mile in Ohms

N Number of conductors per phase

Inductive Reactance

$$X_L = 4\pi f \times 10^{-7} \ln \frac{D_{eq}}{D_{SL}}$$

Where:

X_L Inductive reactance in Ohms/meter

f Frequency of the system in Hertz

D_{eq} Geometric mean distance between phases in meters

D_{SL} Geometric mean radius between conductors of one phase in meters

The **geometric mean distance** between phases is defined as:

$$D_{eq} = \sqrt[3]{d_{ab}d_{bc}d_{ca}}$$

Where:

d_{ab}, d_{bc}, d_{ca} Distances between phases a-b, b-c, c-a, respectively in meters

The **geometric mean radius** between conductors of one phase is defined as:

$$D_{SL} = GMR \quad \text{For 1 stranded conductor}$$

$$D_{SL} = e^{-1/4} r \quad \text{For 1 solid cylindrical conductor}$$

$$D_{SL} = \left(\prod_{k=1}^N \prod_{m=1}^N d_{km} \right)^{\frac{1}{N^2}} \quad \text{For more than 1 conductor bundle}$$

Where:

D_{SL} Geometric mean radius in meters

r External radius of conductor in meters

GMR Geometric mean radius given in tables for one stranded conductor

d_{km} Distance between conductors k and m in meters.

Note: If $k = m$, then $d_{km} = D_{sc}$ for one stranded or solid cylindrical conductor.

Susceptance

$$B = 2\pi f \left(\frac{2\pi\epsilon}{\ln \left(\frac{D_{eq}}{D_{sc}} \right)} \right)$$

Where:

B Susceptance in Siemens/meter

f Frequency of the system in Hertz

ϵ Constant permittivity = 8.85418×10^{-12}

D_{eq} Geometric mean distance between phases, defined as above

D_{sc} Geometric mean radius between conductors of one phase using external radius in meters

The geometric mean radius between conductors of one phase using external radius is defined as:

$$D_{sc} = r \quad \text{For 1 conductor}$$

$$D_{sc} = \left(\prod_{k=1}^N \prod_{m=1}^N d_{km} \right)^{\frac{1}{N^2}} \quad \text{For more than 1 conductor bundle}$$

Where:

D_{sc} Geometric mean radius in meters

r External radius of conductor in meters

d_{km} Distance between conductors k and m in meters.

Note: If $k = m$, then $d_{km} = r$.

Conductance

Assumed $G = 0$

Where:

G Conductance in Siemens/meter

Lumped (Total) Parameters

Resistance, Inductive Reactance, Conductance and Susceptance, using the equivalent π circuit (long line)

$$Z' = R' + jX' = Z_c \sinh \gamma \ell$$

$$Y' = G' + jB' = \frac{2}{Z_c} \tanh \frac{\gamma \ell}{2}$$

Where:

Z' Total series impedance of line in Ohms

Y' Total series admittance of line in Siemens

R' Total series resistance of line in Ohms

X' Total series inductive reactance of line in Ohms

G' Total series conductance of line in Siemens

B' Total series susceptance of line in Siemens

Z_c Characteristic impedance in Ohms

γ Propagation constant in meters⁻¹

ℓ Line length in meters

The characteristic impedance and propagation constant are defined as:

$$Z_c = \sqrt{\frac{z}{y}}$$

$$\gamma = \sqrt{zy}$$

Where:

z Distributed series impedance in Ohms/meter

y Distributed series admittance in Siemens/meter

The **distributed series impedance** and **distributed series admittance** are defined as:

$$z = R + jX$$

$$y = G + jB$$

Where:

- R Distributed series resistance in Ohms/meter
- X Distributed series inductive reactance in Ohms/meter
- G Distributed series conductance in Siemens/meter
- B Distributed series susceptance in Siemens/meter

Surge Impedance Loading

The surge impedance loading is defined as the power delivered by a lossless line to a load resistance equal to the surge (or characteristic) impedance Z_c and is given by:

$$P_{sIL} = \frac{|V_N|^2}{Z_c}$$

Where:

- P_{sIL} Total surge impedance loading in a three-phase line in VA
- V_N Line-line nominal voltage in Volts

Base Values

Impedance Base

$$Z_B = \frac{(V_B^L)^2}{S_B^{3\phi}}$$

Where:

- Z_B Impedance base in Ohms
- $S_B^{3\phi}$ Power base in VA
- V_B^L Line-line voltage base in Volts

Admittance Base

$$Y_B = \frac{1}{Z_B}$$

Where:

Y_B Admittance base in Siemens

Z_B Impedance base in Ohms

Per Unit (PU) Parameters

Resistance, Inductive Reactance, Conductance, Susceptance

$$R_{PU} = \frac{R'}{Z_B}$$

$$X_{PU} = \frac{X'}{Z_B}$$

$$G_{PU} = \frac{G'}{Y_B}$$

$$B_{PU} = \frac{B'}{Y_B}$$

Where:

R_{PU} Per unit resistance

R' Total series resistance in Ohms

X_{PU} Per unit Inductive reactance

X' Total series inductive reactance in Ohms

G_{PU} Per unit conductance

G' Total series conductance in Siemens

B_{PU} Per unit susceptance

B' Total series susceptance in Siemens

Z_B Impedance base in Ohms

Y_B Admittance base in Siemens

MVA To Ampere and Ampere To MVA Limits Conversion

MVA to Ampere Limit Conversion

$$Lim_{Amp} = \frac{Lim_{MVA} \times 10^6}{\sqrt{3} |V_N|}$$

Where:

Lim_{Amp} Limit in Amperes

Lim_{MVA} Limit in MVAs

V_N Nominal voltage in Volts

Ampere to MVA Limit Conversion

$$Lim_{MVA} = Lim_{Amp} \frac{\sqrt{3} |V_N|}{10^6}$$

Where:

Lim_{Amp} Limit in Amperes

Lim_{MVA} Limit in MVAs

V_N Nominal voltage in Volts

Transmission Line/Transformer Options: Transformer Control

Transformers are used to transfer power between different voltage levels or to regulate real or reactive flow through a particular transmission corridor. Most transformers come equipped with taps on the windings to adjust either the voltage transformation or the reactive flow through the transformer. Such transformers are called either load-tap-changing (LTC) transformers or tap-changing-under-load (TCUL) transformers.

Another type of transformer is known as a phase-shifting transformer (phase shifter or phase angle regulator). Phase-shifting transformers, which are less common than LTC transformers, vary the angle of the phase shift across the transformer in order to control the MW power flow through the transformer.

Off-nominal Turns Ratio and Phase Shift Degrees

The [Line/Transformer Dialog](#) displays several transmission line and transformer properties. For transformers, this dialog box also shows information about the LTC or phase shifter controls. The **Off-nominal Turns Ratio** field indicates the voltage transformation, while the **Phase Shift Degrees** field show the phase shift angle. If the transformer is not on automatic control, these values can be changed manually. The off-nominal tap ratio determines the additional transformation relative to the nominal transformation. This value normally ranges from 0.9 to 1.1 (1.0 corresponds to no additional transformation). For phase-shifting transformers the phase shift value normally ranges from about -40° to 40° . The phase angle field can be non-zero for LTC and fixed transformers, most notably $\pm 30^\circ$ if the transformer configuration is a delta-wye or wye-delta configuration. The transformer configuration is very important when performing a [fault analysis](#) study.

When in Edit Mode, the dialog also reveals the type of transformer. Valid types are 1) No Automatic Control (in which the taps are assumed fixed), 2) voltage regulation (AVR), 3) Reactive Power Control, and 4) Phase Shift Control. The type of transformer CANNOT be modified in the Run Mode.

Simulator provides you with a great deal of flexibility in being able to specify which transformers will actually be used for automatic control in the Power Flow Solution. For a transformer to be used for voltage or flow control, three criteria must be met.

- The transformer's **Automatic Control Enabled** field must be checked on its [Line/Transformer dialog](#). This field can also be modified on the [Transformer Records display](#).
- The transformer's area must have automatic transformer control enabled. This is specified on the *Options Tab* of the [Area Records](#) display.
- Transformer control must not be disabled for the entire case. This is specified on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#).

The area and case enforcement of transformer control are also accessible from the Run Mode [Line/Transformer Dialog](#).

Automatic Control

The **Automatic Control** fields are only visible in Edit Mode when the [Change Automatic Control Options](#) button is pressed.

No Automatic Control

On this control setting the transformer will operate at the given off-nominal turns ratio and phase shift, and will remain fixed at those values during the entire solution process unless manually changed by the user.

AVR (Automatic Voltage Regulation)

When on automatic voltage control, the transformer taps automatically change to keep the voltage at the regulated bus (usually one of the terminal buses of the transformer) within a voltage range between the minimum voltage and maximum voltage values (given in per unit). These values can be seen by clicking on the [Automatic Control Options](#) button. Note that automatic control is possible only if a regulated bus has been specified.

The tap position for an LTC transformer is indicated on the oneline by the number of tap step positions from the nominal position (i.e., the position when the off-nominal tap ratio is equal to 1.0). When the off-nominal ratio is greater than 1.0, the transformer's tap is said to be in the "raise" position, and an 'R' appears after the number. Likewise, when the off-nominal ratio is less than 1.0, the transformer's tap is said to be in the "lower" position, and an 'L' appears after the number. For example, with a step size of 0.00625 and an off-nominal ratio of 1.05, the tap would be in position 8R. The tap position can be changed manually only when the transformer has been set off automatic voltage control.

Simulator will also detect instances when controlling transformers are in parallel, and will employ checks during the solution routine to prevent the controllers from fighting each other and potentially going to opposite tap solutions, which could result in unwanted loop flow through the transformer objects. This option is enabled by default, but can be turned off in the [Power Flow Solution General Options](#).

Transformer Reactive Power Control

When on automatic reactive power control, the transformer taps automatically change to keep the reactive power flow through the transformer (measured at the *from bus*) within a user-specified range. The reactive power control parameters can be seen by clicking the [Automatic Control Options](#) button.

Phase Shift Control

When a transformer is on phase shift control, the transformer phase shift angle automatically changes to keep the MW flow through the transformer (measured at the *from bus*) between the minimum and maximum flow values (given in MW, with flow into the transformer assumed positive). The limits on the phase shifting angles are specified in the minimum and maximum phase fields (in degrees). These values can be seen by clicking on the [Automatic Control Options](#) button. The phase shift angle changes in discrete steps, with the step size specified in the Step Size field (in degrees). The **MW Per Phase Angle Step Size** provides an estimate of the change in the controlled MW flow value if the phase angle is increased by the step size value.

Specify Transformer Bases and Impedances

Shows the Transformer Bases and Impedances Dialog. This dialog allows the user to specify the transformer parameters in per unit on the transformer base (taken as its rating). Click **OK** to convert all the transformer parameters values to the system base specified in the [General Power Flow Solution Options](#).

Transformer AVR Dialog

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this display, click the *Automatic Control Options* button on the [Line/Transformer Information Dialog](#).

This dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control.

Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

Present Regulated Bus Voltage

The present voltage of the regulated bus.

Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the *Minimum Voltage* and *Maximum Voltage* fields), the *Voltage Error* field indicates by how much the voltage deviates from the control range.

Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

Present Tap Ratio

The tap ratio of the transformer for the current system state.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on *Power Flow Solution Tab* of the [Simulator Options dialog](#).

Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transmission Line/Transformer Options: Series Capacitor

The Series Capacitor tab of the Line/Transformer Information dialog displays information relating to a series capacitor, including its status.

Status

The capacitor itself has two status positions, **Bypassed** and **In Service**. When the series capacitor is in service, the branch is modeled as a reactive branch, using the line parameters from the Parameters / Display page. If the capacitor is bypassed, a low impedance branch is introduced to bypass the capacitor. Note that this is not the same as the branch status of **Open** or **Closed**. The branch status is the indicator of whether or not the entire circuit is operating (closed), regardless of **Bypassed** or **In Service** status on the capacitor itself.

Is Series Capacitor

If this box is checked, the branch can be treated as a series capacitor, with the series Status of Bypassed or In-Service available.

Multi-Section Line Information

Multi-section line records are used to group a number of series-connected transmission lines together so that their status is controlled as a single entity. They are usually used to model very long transmission lines that require multiple individual transmission line records if they are to be modeled accurately. In terms of line status, Simulator then treats each multi-section line as a single device. In other words, changing the status of one line in the record changes the status of the other lines in the record, as well.

Multi-section line records are **not** directly represented on the oneline diagrams. Rather, to view this dialog, right-click the record of interest on the [Multi-Section Line Records](#) display and select **Show Dialog** from the resulting local menu. To define a new multi-section line, switch to Edit Mode and, from the Multi-Section Line Records display, right-click and select **Insert** from the resulting local menu.

Note: If any of the transmission lines comprising a multi-section line record are deleted, the entire multi-section line record is deleted as well.

The Multi-Section Lines Dialog contains the following fields:

From Bus Number and Name

Number and name of the from bus for the record. If you are defining a new multi-section line, this must be the first data item you specify.

To Bus Number and Name

Number and name of the "to" bus for the record. You cannot specify this value directly. Instead, use the table at the bottom of the dialog to define the intermediate buses and *to* terminal.

Circuit

Two character identifier used to distinguish between multiple records joining the same from/to buses. The first character of the circuit identifier must be an "&."

You can use the spin button immediately to the right of the circuit field to view other multi-section line records. However if you have changed the record, **you must** select **Save** before moving to another record. Otherwise, your changes will be lost.

From End Metered

If checked, the *from end* of the record is the metered end; otherwise the *to end* is metered.

Multi-Section Line Name

The name for the multi-section line.

Table

The table lists the dummy (or intermediate) buses and the *to bus* that comprise the record. The first column should contain the first dummy bus number and the circuit ID of the transmission line joining the *from bus* with the first dummy bus. The next column should contain the second dummy bus number and the circuit id of the transmission line joining this bus to the first dummy bus. Continue until the last column contains the *to* bus and the circuit ID of the transmission line joining the last dummy bus with the "to" bus.

Each transmission line comprising the multi-section line must already exist. If only one transmission line joins any two buses, you may omit the circuit identifier.

OK

Saves any modifications and closes the dialog.

Save

Saves any modifications but does not close the dialog.

Cancel

Closes the dialog without saving modifications to the current record.

Delete

Deletes the current multi-section line record.

Note: If any of the transmission lines comprising a multi-section line record are deleted, the entire multi-section line record is deleted as well.

Line Field Options

Line field objects are used to show different values associated with transmission lines, transformers and DC lines. This dialog is used to view and modify the parameters associated with these fields.

Find...

If you do not know the exact line identifiers you are looking for, you can click this button to open the [advanced search engine](#).

Near Bus Number

Bus associated with the near end of the object. All fields display values calculated at the *near bus* end. When inserting fields graphically, this field is automatically set to the closest bus on the oneline.

Far Bus Number

Bus associated with the *far end* of the object.

Circuit

Two-character identifier used to distinguish between multiple lines or transformers joining the same two buses. Default is '1'.

Total Digits in Fields

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Delta Per Mouse Click

Only used with the MVA Limit field type. Specifying a nonzero value for this field equips the MVA Limit Field with an integrated spin button that can be clicked to increment or decrement the MVA Limit by the amount of the *Delta Per Mouse Click* value.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Anchored

If checked, the line analog is [anchored](#) to its associated line.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of line field to show. The following choices are available:

AC Line MW Flow MW flow into the line or transformer at the near bus

AC Line Mvar Flow Mvar flow into the line or transformer at the near bus

AC Line MVA Flow Magnitude of MVA flow into the line or transformer at the near bus

AC Line Amp Flow Magnitude of Amp flow (in amps) into the line at the near bus

AC Line MW Losses Real power losses on the line or transformer in MW

AC Line Mvar Losses Reactive power losses on the line or transformer in Mvar

DC Line MW Flow MW flow into the DC line at the near bus

DC Line Mvar Flow Mvar flow into the DC line at the near bus

MVA Limit MVA limit for the line or transformer

DC Line Set point Set point value for the DC line; see [DC Transmission Line dialog](#)

Select a Field Choose from over 80 different fields

Select **OK** to save changes and close the dialog or **Cancel** to close the dialog without saving your changes.

Series Capacitor Field Options Dialog

Series capacitor field objects are used to show field values specific to series capacitors. Use [Line Fields](#) to show fields generic to transformers, series capacitors and transmission lines, such as the flow of power through the device. The series capacitor fields dialog is used to view and modify the parameters associated with series capacitor specific fields.

Near Bus Number

Bus associated with the *near end* of the series capacitor.

Far Bus Number

Bus associated with the *far end* of the series capacitor.

Circuit

Two-character identifier used to distinguish between branches joining the same two buses. Default is '1'.

Anchored

When checked, the text field will move with the series capacitor if the series capacitor is moved on the oneline diagram.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The angle at which the text will be placed.

Field Value

The value of the currently selected field.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Designates the type of transformer field to show. The following choices are available:

Status Capacitor status

Series Capacitance Capacitance of series capacitor

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

Transformer Field Options Dialog

Transformer field objects are used to show field values specific to transformers. Use [Line Fields](#) to show fields generic to transformers and transmission lines, such as the flow of power through the device. The transformer fields dialog is used to view and modify the parameters associated with transformer-specific fields.

Near Bus Number

Bus associated with the *near end* of the transformer.

Far Bus Number

Bus associated with the *far end* of the transformer.

Circuit

Two-character identifier used to distinguish between transformers joining the same two buses. Default is '1'.

Find...

If you do not know the exact transformer you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Delta per Mouse Click

This value is used only with the **Off-nominal Tap Ratio** and **Phase Shift Angle** field types. When there is a nonzero entry in this field, and the field type is valid, a spin button is shown to the right of the zone field. When the up spin button is clicked, the field value is increased by this number; when the down button is clicked, the field value is decreased by this amount.

Field Value

Shows the current output for the transformer field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Anchored

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Designates the type of transformer field to show. The following choices are available:

Off-nominal Tap Ratio Actual tap ratio

Phase Shift Angle Actual phase shift in degrees

Off-nominal Tap Position Tap position in steps, usually ranging from L16 to R16

Automatic Control Status The status of the control for the transformer

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

DC Transmission Line Options

This dialog is used to view and modify the parameters associated with each two-terminal DC transmission line in the system. It can also be used to insert new two-terminal DC transmission lines or to delete existing ones.

This dialog has six separate pages: [Line Parameters](#), [Rectifier Parameters](#), [Inverter Parameters](#), [Actual Flows](#), [OPF](#) and [Custom](#). The Actual Flows page is not used in Edit Mode. The separate pages can be accessed using the tabs shown at the top of the dialog. These pages can be used to view/change the modeling parameters associated with the DC lines.

DC Line Options: Line Parameters

This page is used to enter parameters associated with the DC line itself.

Rectifier and Inverter Bus Numbers and Names

These fields indicate the numbers and names of the rectifier and inverter ends of the line. When graphically inserting a DC line, these fields are automatically determined based upon the starting and ending buses used in drawing the line. When investigating existing DC line records, you may use the *Find By Numbers* button to identify a DC line between a specific rectifier - inverter pair. You may also use the spin control to cycle through the list of DC line records modeled in the case.

Circuit ID

This field is not currently used, since only one DC transmission line can exist between each inverter-rectifier pair. It is hard-coded to be 1.

Find By Numbers

To find a DC line by its bus numbers, enter the *Rectifier* and *Inverter* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of DC lines in the system.

Area Name

Names of the areas in which the rectifier and inverter buses are located.

Labels

Clicking this button will open the [Subscribed Aliases dialog](#) displaying the list of defined labels for the DC line. New labels can also be added for the DC Line from the dialog as well.

Link to New DC Line

Use the *Link to New DC Line* button to create a new DC line corresponding to the entries you have made in the dialog. This button performs the same function as pressing *Save*. Note that adding a new DC line to the case in this way does not add a DC line to the online display; the new DC line is present only in the model. You may then add the newly modeled DC line to the online diagram in the usual way (such as select **Network > DC Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab).

Line Parameters

Status

Operating status of the DC transmission line. If the status is Open, then no power can flow on the DC line regardless of the control mode setting. If the status is closed, then the control mode will dictate how power flows on the line.

Control Mode

The initial control mode for the line. Specify *Blocked* to disable the DC line, *Power* to maintain specified MW power flow through the line, or *Current* to maintain specified current flow through the line.

Setpoint

If the line operates in the Power Control Mode, *Setpoint* should indicate the desired power flow in MW. To specify power flow at the rectifier end, enter a positive value. Enter a negative value to

specify power flow at the inverter end. If the DC line operates in Current Control Mode, enter the desired line flow in amps.

Resistance

Resistance of the DC Line.

Sched. Voltage

Scheduled DC line voltage in kV. The value of Rcomp is used to determine whether this value specifies the inverter end or the rectifier end.

Switch Voltage

When the line operates in the Power Control Mode, this is the inverter voltage level in kV at which the line switches from constant power to constant current control.

RComp

Compounding resistance. The compounding resistance dictates whether the inverter voltage is scheduled ($R_{comp} = 0$), or the rectifier voltage is scheduled ($R_{comp} = \text{DC line resistance}$). Simulator does not currently support scheduling the voltage at a point along the DC line ($0 < R_{com} < \text{DC line resistance}$), but instead will treat any non-zero value of Rcomp as scheduling the voltage at the rectifier.

Setpoint Specified At

Indicates which end of the DC transmission line the Setpoint value is designated. This will be the terminal where the setpoint value is maintained. The opposite terminal flow value will be a calculated quantity.

Metered End of Line

Indicates which end of the DC line is assumed metered for Area interchange calculations.

DC Line Options: Rectifier Parameters

This page is used to enter parameters associated with the rectifier end of the line.

of Bridges

Number of valve bridges in series.

Base Voltage

Base AC voltage in kV on primary side of transformer.

XF Ratio

Transformer ratio.

XF Tap

Transformer tap setting.

XF Min/Max Tap, XF Tap Step

Transformer minimum and maximum tap settings, and the tap's step size.

Commuting XF Resistance and Reactance

Commuting resistance and reactance for the transformer, in ohms.

Minimum, Maximum, and Actual Firing Angle

Minimum, maximum, and actual values of the firing angle for the rectifier.

DC Line Options: Inverter Parameters

This page is used to enter parameters associated with the inverter end of the line. Entries are identical to the Rectifier Page, except here they are associated with the inverter.

of Bridges

Number of valve bridges in series.

Base Voltage

Base AC voltage in kV on primary side of transformer.

XF Ratio

Transformer ratio.

XF Tap

Transformer tap setting.

XF Min/Max Tap, XF Tap Step

Transformer minimum and maximum tap settings, and the tap's step size.

Commuting XF Resistance and Reactance

Commuting resistance and reactance for the transformer, in ohms.

Minimum, Maximum, and Actual Firing Angle

Minimum, maximum, and actual values of the firing angle for the rectifier.

DC Line Options: Actual Flow

Shows the actual real and reactive power flows into the line at the rectifier and the inverter, as well as the voltage at both ends (in kV) and the line current in amps. The contents of this page are valid only when Simulator is in Run Mode.

DC Line Options: OPF

This page is used to enter parameters associated with the OPF.

Rectifier/Inverter Bus MW Marginal Costs

Displays the marginal costs of the branches terminal buses, following the solution of the OPF.

OPF Control**Minimum and Maximum Setpoint Value**

The minimum and maximum setpoint allowed for the DC Line. When on OPF Control the DC Line will be dispatched between these values.

Current Setpoint Value

The current Setpoint as found on the Line Parameters Tab.

Transmission Charge

This is the cost incurred for moving power across the DC transmission line. The cost is zero at a zero MW flow and increases linearly using this charge.

OPF Control Enabled for this DC line

This check box must be checked if the DC limit is going to be enforced when running an OPF solution. If this box is not checked, the OPF routine will allow the DC line to violate its limits.

Include Impact of DC Line Marginal Losses

When dispatching the DC Line this determines whether the marginal losses from the DC line are taken into account.

Multi-Terminal DC Record Information

This dialog is used to view and modify the parameters associated with multi-terminal DC records. It is also used when inserting new multi-terminal DC records.

The Multi-Terminal DC Record dialog can be used to inspect and modify the model of a multi-terminal DC network record. To view the Multi-Terminal DC Record, simply right-click on the record of interest in the [Multi-Terminal DC Record Display](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

Record Number

Unique number between 1 and 999 which identifies the current multi-terminal DC record.

Number of Devices

Lists the number of DC buses, converters, and DC Lines that form the multi-terminal DC network.

Control

The control method used when solving the multi-terminal DC network.

Controlling Converter

The AC converter bus number where the DC voltage is being controlled.

MTDC Network Status

Status of the entire Multi-terminal DC network. If this field is set to Closed, the entire DC subnetwork of the MTDC model is considered disconnected in the load flow case.

DC Buses Tab

This page of the dialog displays the DC bus records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC bus, right-click on the record of interest and choose *Show Dialog* from the popup menu.

DC Converters Tab

This page of the dialog displays the DC converter records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC converter, right-click on the record of interest and choose *Show Dialog* from the popup menu.

DC Lines Tab

This page of the dialog displays the DC line records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC line, right-click on the record of interest and choose *Show Dialog* from the popup menu.

Custom Tab

Enter any text notes you wish in the Memo page. When the case is saved as a Simulator PWB file, the memo text will also be saved. Custom strings and values can also be entered and saved with the case file as well.

Multi-Terminal DC Bus Information

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

DC Bus Number, DC Bus Name

The number and name identifiers for the selected DC bus.

AC Bus Number

The AC bus number connected to the DC bus through an AC/DC converter. If the selected DC bus is a bus that is internal to the DC multi-terminal network (not directly connected to an AC bus,) this field will be 0.

Area Number, Area Name

The number and name identifiers of the control area the DC bus is contained in.

Zone Number, Zone Name

The number and name identifiers of the zone the DC bus is contained in.

Ground Resistance

Resistance to ground of the DC bus, entered in Ohms. This field is currently only for storage of values supported by other load flow formats, and is currently not used by Simulator.

Multi-Terminal DC Converter Information

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

Converter Parameters

Number of Bridges

Number of bridges in series for the selected converter.

Converter Type

R for rectifier or **I** for inverter.

Commutating Impedance

Commutating impedance per bridge, in Ohms.

Firing Angle Limits

The maximum and minimum firing angle limits, in degrees.

Transformer Parameters

AC Base

The primary AC base voltage, in kV.

DC Base

The DC base voltage, in kV.

Transformer Ratio

Actual transformer ratio.

Tap Settings

Displays the actual tap setting, the tap step, and maximum and minimum tap values for the converter transformer.

Control Parameters

Setpoint

The setpoint control value at the converter. For the voltage-controlling converter, this field is set to 0. For the remaining converters, this field displays MW when in Power mode, or Amps when in Current mode.

Margin

Rectifier margin, entered in per-unit of the DC power or current. This field is currently only for support of other load flow formats, and is not used by Simulator.

DC Participation Factor

Converter participation factor. This field is currently only for support of other load flow formats, and is not used by Simulator.

Voltage

The DC Voltage magnitude at the DC side of the converter.

Solved Parameters

Firing Angle

The firing angle of the converter, as determined during the load flow solution.

DC Current

The calculated DC current at the converter DC terminal.

MW, MVAR

The real and reactive power delivered to (or absorbed from) the AC system by the converter.

Multi-Terminal DC Line Information

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

From and To DC Bus Number

The DC bus numbers of the From and To buses in the multi-terminal DC network. These fields must contain valid DC bus numbers of the selected multi-terminal DC record. AC bus numbers from the load flow case are not acceptable bus numbers for a multi-terminal DC line.

DC Circuit

The circuit identifier for the DC line.

DC Resistance and DC Inductance

The resistance and inductance of the DC line. Resistance is in Ohms, and is used for solving the load flow of the DC network. The inductance is in milliHenries, and is not used for solving the load flow. The inductance field is currently only for support of other load flow formats, and is not used by Simulator.

Transmission Line/Transformer Options: Transformer Control

Transformers are used to transfer power between different voltage levels or to regulate real or reactive flow through a particular transmission corridor. Most transformers come equipped with taps on the windings to adjust either the voltage transformation or the reactive flow through the transformer. Such transformers are called either load-tap-changing (LTC) transformers or tap-changing-under-load (TCUL) transformers.

Another type of transformer is known as a phase-shifting transformer (phase shifter or phase angle regulator). Phase-shifting transformers, which are less common than LTC transformers, vary the angle of the phase shift across the transformer in order to control the MW power flow through the transformer.

Off-nominal Turns Ratio and Phase Shift Degrees

The [Line/Transformer Dialog](#) displays several transmission line and transformer properties. For transformers, this dialog box also shows information about the LTC or phase shifter controls. The **Off-nominal Turns Ratio** field indicates the voltage transformation, while the **Phase Shift Degrees** field show the phase shift angle. If the transformer is not on automatic control, these values can be changed manually. The off-nominal tap ratio determines the additional transformation relative to the nominal transformation. This value normally ranges from 0.9 to 1.1 (1.0 corresponds to no additional transformation). For phase-shifting transformers the phase shift value normally ranges from about -40° to 40° . The phase angle field can be non-zero for LTC and fixed transformers, most notably $\pm 30^\circ$ if the transformer configuration is a delta-wye or wye-delta configuration. The transformer configuration is very important when performing a [fault analysis](#) study.

When in Edit Mode, the dialog also reveals the type of transformer. Valid types are 1) No Automatic Control (in which the taps are assumed fixed), 2) voltage regulation (AVR), 3) Reactive Power Control, and 4) Phase Shift Control. The type of transformer CANNOT be modified in the Run Mode.

Simulator provides you with a great deal of flexibility in being able to specify which transformers will actually be used for automatic control in the Power Flow Solution. For a transformer to be used for voltage or flow control, three criteria must be met.

- The transformer's **Automatic Control Enabled** field must be checked on its [Line/Transformer dialog](#). This field can also be modified on the [Transformer Records display](#).
- The transformer's area must have automatic transformer control enabled. This is specified on the *Options Tab* of the [Area Records](#) display.
- Transformer control must not be disabled for the entire case. This is specified on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#).

The area and case enforcement of transformer control are also accessible from the Run Mode [Line/Transformer Dialog](#).

Automatic Control

The **Automatic Control** fields are only visible in Edit Mode when the [Change Automatic Control Options](#) button is pressed.

No Automatic Control

On this control setting the transformer will operate at the given off-nominal turns ratio and phase shift, and will remain fixed at those values during the entire solution process unless manually changed by the user.

AVR (Automatic Voltage Regulation)

When on automatic voltage control, the transformer taps automatically change to keep the voltage at the regulated bus (usually one of the terminal buses of the transformer) within a voltage range between the minimum voltage and maximum voltage values (given in per unit). These values can be seen by clicking on the [Automatic Control Options](#) button. Note that automatic control is possible only if a regulated bus has been specified.

The tap position for an LTC transformer is indicated on the oneline by the number of tap step positions from the nominal position (i.e., the position when the off-nominal tap ratio is equal to 1.0). When the off-nominal ratio is greater than 1.0, the transformer's tap is said to be in the "raise" position, and an 'R' appears after the number. Likewise, when the off-nominal ratio is less than 1.0, the transformer's tap is said to be in the "lower" position, and an 'L' appears after the number. For example, with a step size of 0.00625 and an off-nominal ratio of 1.05, the tap would be in position 8R. The tap position can be changed manually only when the transformer has been set off automatic voltage control.

Simulator will also detect instances when controlling transformers are in parallel, and will employ checks during the solution routine to prevent the controllers from fighting each other and potentially going to opposite tap solutions, which could result in unwanted loop flow through the transformer objects. This option is enabled by default, but can be turned off in the [Power Flow Solution General Options](#).

Transformer Reactive Power Control

When on automatic reactive power control, the transformer taps automatically change to keep the reactive power flow through the transformer (measured at the *from bus*) within a user-specified range. The reactive power control parameters can be seen by clicking the [Automatic Control Options](#) button.

Phase Shift Control

When a transformer is on phase shift control, the transformer phase shift angle automatically changes to keep the MW flow through the transformer (measured at the *from bus*) between the minimum and maximum flow values (given in MW, with flow into the transformer assumed positive). The limits on the phase shifting angles are specified in the minimum and maximum phase fields (in degrees). These values can be seen by clicking on the [Automatic Control Options](#) button. The phase shift angle changes in discrete steps, with the step size specified in the Step Size field (in degrees). The **MW Per Phase Angle Step Size** provides an estimate of the change in the controlled MW flow value if the phase angle is increased by the step size value.

Specify Transformer Bases and Impedances

Shows the Transformer Bases and Impedances Dialog. This dialog allows the user to specify the transformer parameters in per unit on the transformer base (taken as its rating). Click **OK** to convert all the transformer parameters values to the system base specified in the [General Power Flow Solution Options](#).

Transformers Bases and Impedances Dialog

Typically the impedances and tap values of transformers is already assumed to have been converted to unity tap base and bus nominal voltage base. However, some load flow formats provide the taps and impedances on specific transformer bases, which are different than the bus voltage and unity tap base assumptions. In these cases, Simulator will convert parameters from the transformer bases to the unity tap and bus nominal voltage base. Display of the impedances and tap values normally displayed in the [Line/Transformer Options](#) dialog are displayed on the Simulator assumed bases. However, if you wish to view the original transformer values on the transformer supplied bases, this dialog will display the original values. You can modify the original values stored here in this dialog. Note that when you do so, the converted values that Simulator stores on the system bases will also be automatically updated to reflect the change that has been made to the original values on the transformer bases.

Transformer Impedance Correction Table Display

The Transformer Impedance Correction Display shows information about all the transformer impedance correction tables in the case. The Transformer Impedance Correction Display is used to model the change in the impedance of the transformer as the off-nominal turns ratio or phase shift angle is varied.

The Correction Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated correction tables. When in Edit Mode, you can define new tables using the *Insert* option, or delete existing tables using *Delete*. You can also sort the transformer impedance correction information by clicking on the heading of the field by which you want to sort.

To show this display select **Network > Impedance Correction Tables** from the [Model Explorer](#).

The display contains the following fields by default:

Table #, Table Name

Shows the table number and name for the record. Table number must be between 1 and 64.

Transformer Impedance Scaling Factors

The rest of the default columns show the actual fields in the table. The first field of each pair shows the off-nominal turns ratio or phase shift angle, while the second field shows the associated scaling factor for the transformer's impedance.

Transformer AVR Dialog

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this display, click the *Automatic Control Options* button on the [Line/Transformer Information Dialog](#).

This dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control.

Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

Present Regulated Bus Voltage

The present voltage of the regulated bus.

Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the *Minimum Voltage* and *Maximum Voltage* fields), the *Voltage Error* field indicates by how much the voltage deviates from the control range.

Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

Present Tap Ratio

The tap ratio of the transformer for the current system state.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on *Power Flow Solution Tab* of the [Simulator Options dialog](#).

Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transformer Mvar Control Dialog

The Transformer Mvar Control dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers that are used to control the Mvar flow through the transformer. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#).

When used to control reactive power, the LTC transformer always controls the reactive power flow at the *from* end of the transformer (i.e., the tapped side), with positive flow assumed to be going through the transformer to the *to* bus. Therefore the regulated bus field is not used.

The dialog has the following fields:

Mvar Flow at *From* Bus

The current Mvar flow as measured at the *from* end of the line. This is the parameter the transformer tries to control.

Mvar Error

If the Mvar flow at the *from* end violates the limits defined by the *Minimum Mvar Flow* and *Maximum Mvar Flow* fields, the *Mvar Error* field indicates by how much the flow falls outside the control range.

Regulation Minimum Mvar Flow, Regulation Maximum Mvar Flow

Minimum and maximum allowable reactive power flow as measured at the *from bus*. The transformer attempts to regulate the reactive flow to fall within this range.

Regulation Target Type

As long as the regulated Mvar flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated Mvar flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated Mvar flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the Mvar flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated Mvar flow is above the maximum, and regulation minimum is used as the target value when the regulated Mvar flow is below the minimum.

Present Tap Ratio

The transformer's present off-nominal turns ratio.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Mvar to Tap Sensitivity

The amount of Mvar shift that would be implemented by switching one tap position from the current position. This sensitivity indicates the ability of the transformer to control Mvars.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transformer Phase Shifting Information

The Transformer Phase Shifting Dialog is used to view the control parameters of phase-shifting transformers. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#), provided that the *Phase Shift Control* option is chosen from the *Automatic Control* group.

Regulated Bus Number

Number of the terminal bus of the phase shifter regulated by the phase shifter. When control is active, the phase shifter will automatically change its phase shift to keep the MW flow at this bus at the desired value.

Current MW Flow

Current MW flow through the transformer measured at the regulated bus terminal.

MW Error

If the current MW flow falls outside the minimum/maximum MW flow limits, the *MW Error* field indicates by how much the flow violates the regulating range.

Regulation Minimum MW Flow, Regulation Maximum MW Flow

Minimum and maximum allowable MW flow through the phase shifter.

Regulation Target Type

As long as the regulated MW flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated MW flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated MW flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the MW flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated MW flow is above the maximum, and regulation minimum is used as the target value when the regulated MW flow is below the minimum.

Present Phase Angle (Degrees)

The phase angle of the transformer for the current solved system state.

Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

Step Size (Degrees)

Phase shift change per step in degrees.

MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an

existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transformer Field Options Dialog

Transformer field objects are used to show field values specific to transformers. Use [Line Fields](#) to show fields generic to transformers and transmission lines, such as the flow of power through the device. The transformer fields dialog is used to view and modify the parameters associated with transformer-specific fields.

Near Bus Number

Bus associated with the *near end* of the transformer.

Far Bus Number

Bus associated with the *far end* of the transformer.

Circuit

Two-character identifier used to distinguish between transformers joining the same two buses. Default is '1'.

Find...

If you do not know the exact transformer you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Delta per Mouse Click

This value is used only with the **Off-nominal Tap Ratio** and **Phase Shift Angle** field types. When there is a nonzero entry in this field, and the field type is valid, a spin button is shown to the right of the zone field. When the up spin button is clicked, the field value is increased by this number; when the down button is clicked, the field value is decreased by this amount.

Field Value

Shows the current output for the transformer field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Anchored

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Designates the type of transformer field to show. The following choices are available:

Off-nominal Tap Ratio Actual tap ratio

Phase Shift Angle Actual phase shift in degrees

Off-nominal Tap Position Tap position in steps, usually ranging from L16 to R16

Automatic Control Status The status of the control for the transformer

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

Three Winding Transformer Information

The Three Winding Transformer Dialog is used to create, modify or delete three winding transformer records in Edit Mode, or to view information for a specific three winding transformer record in run mode. Note that all the values displayed on this dialog are on the system MVA base. If the records were created them from a file, the values are automatically converted to the system base. If a three winding transformer record is entered manually, the parameters need to be entered into Simulator computed with the same system MVA base Simulator is using.

This dialog has the following controls:

Primary Winding

This section of the dialog displays the primary winding terminal bus number, nominal kV, and fixed tap value (in per unit). In addition, the automatic tap changer is assumed to be on the primary winding of a three winding transformer. Therefore the LTC field displays the tap changer tap value (in per unit) on the primary winding.

Secondary Winding

This section of the dialog displays the secondary winding terminal bus number, nominal kV, and fixed tap value (in per unit).

Tertiary Winding

This section of the dialog displays the tertiary winding terminal bus number, nominal kV, and fixed tap value (in per unit).

Star Bus (Internal Node)

This section of the dialog displays the internal node parameters of the three winding transformer model. Three winding transformers are modeled as three two winding transformers connected at the three winding transformer terminal buses to a common or internal node, referred to as the star bus. The parameters displayed for the star bus are the bus number, voltage (in per unit), and angle.

Primary-Secondary, Secondary-Tertiary, and Tertiary-Primary Impedance

These are the actual three winding transformer winding to winding impedances, in per unit on the system base. These values are used to compute the equivalent two winding transformer impedances for the two winding transformers used to model the three winding transformer operation.

Transformer Parameters

The values for the magnetizing conductance (G) and magnetizing susceptance (G), in per unit on the system base.

Circuit ID

The circuit identifier for the three winding transformer.

Status

The status of the three winding transformer. If checked, the three winding transformer model is in service, otherwise the equivalent model is treated as out of service.

Mathematically equivalent two-winding transformers

This table displays the three two winding transformers that are mathematically equivalent representations of the three winding transformer. If you read the three winding transformer record from a file, the two winding equivalent transformers are created automatically. If you are inserting a three winding transformer manually, you can set the parameters for the primary, secondary and

tertiary windings in the fields above, then click the **Set Two-Winding Equivalent Transformers** button to have Simulator automatically create the two winding transformer records for you. You can also right-click in this table and insert, modify or delete two winding transformers manually if you already have the two winding transformer representations created.

Once you are finished with the dialog, you can click **Save** or **OK** to save any changes. If you wish to abandon any changes you have made, click **Cancel**.

Switched Shunt Information (Edit Mode)

This dialog is used to view and modify the parameters associated with each switched shunt in the system. It can also be used to insert new switched shunts and sometimes to delete existing shunts. Only one switched shunt is permitted at each bus. Switched shunts usually consist of either capacitors to supply reactive power (in MVR) to the system, or reactors to absorb reactive power. The switched shunts are represented by a number of blocks of admittance that can be switched in a number of discrete steps. If at least one block is in service, the shunt is said to be online. The shunt's corresponding circuit breaker can be used to determine and to toggle the switched shunt's status.

The Edit Mode version of this dialog is very similar in content to its [Run Mode](#) counterpart.

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the switched shunt is attached. You can use the spin button immediately to the right of the number to move to the next switched shunt (click the up arrow) or the previous switched shunt (click the down arrow).

Find By Number

To find a switched shunt by its bus number, enter the number into the Bus Number field. Then click this button.

Bus Name

Unique alphabetic identifier for the bus to which the switched shunt is attached, consisting of up to eight characters.

Find By Name

To find a switched shunt by its bus name, enter the bus name into the Bus Name field (case insensitive). Then click this button.

Shunt ID

Since multiple switched shunts are allowed on a single bus, each switched shunt has a unique Shunt ID.

Find...

If you do not know the exact switched shunt bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Open or closed status of the switched shunt.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected load.

Area Number, Area Name

The area number and name to which the switched shunt belongs. Note that you can change the area of the switched shunt to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the switched shunt belongs. Note that you can change the zone of the switched shunt to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different zone than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The name and number of the substation to which the switched shunt belongs, and it is the same than that of the terminal bus.

Display

Display Size

Size of the switched shunt.

Scale Width with Size

Automatically scales the width of the symbol when the object is resized.

Display Width

Width of the switched shunt symbol.

Pixel Thickness

Thickness of the display object in pixels.

Orientation

Specifies the direction in which to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

Link to New Shunt

Adds a new record in the data or links the selected shunt to a different record.

OK, Save, Delete, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the Find By Number command to edit additional switched shunts. **Delete** deletes the current switched shunt; this option is not available when inserting objects graphically – use the cut command instead. **Cancel** closes the dialog but does not save any changes.

Parameters

Nominal Mvar

The Nominal Mvar field gives the initial amount of reactive power the device would supply (in Mvars) if its terminal voltage were 1.0 per unit.

Nominal MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. In that case, it is possible for the bus shunt to have both a MW and MVAR component. The MW component will be displayed here. In general, switched shunts of other control types do not have MW components, and this field will not be displayed. If you change a switched shunt read as a bus shunt to another form of control, the MW component will remain, but has no controllability.

Control Mode

Determines whether the switched shunt has a fixed value, or whether the amount of reactive power supplied by the device changes in either discrete steps or continuously in order to maintain its terminal voltage within the voltage range specified in the Voltage Regulation fields. This field can be changed (except in Viewer). However, for a switched shunt to be used for automatic control, three fields must be set correctly: 1) the *Control Mode* field must be set to either *Discrete* or *Continuous*, 2) the corresponding area's *Auto Shunts* property must be true, and 3) the case-wide *Disable Switched Shunt Control* option, which can be set on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#), must not be checked.

Note: automatic control of switched shunts is disabled if the voltage regulation high value is not greater than the low value; they should not be equal unless in the continuous mode.

Note the additional control mode called Bus Shunt (Fixed). This is analogous to the shunt MW and MVAR values that can also be stored at the bus level. The difference is that bus shunts stored directly with the bus cannot be turned on and off in the load flow; rather they are always included in the load flow solution. Bus shunts that are represented as switched shunt objects on Bus Shunt control, however, are mathematically exactly the same and can be turned on or off. The reason for the differentiation of the Bus Shunt versus normal Fixed control is that the Bus Shunt control type is intended to identify the difference between a bus shunt and a transformer that MAY have controllability, but is currently turned off control by being set to a Fixed value.

Control Regulation Settings

Voltage Regulation

When the switched shunt is on automatic control, its reactive power is changed in discrete steps or continuously to keep the voltage at the regulated bus within the per unit voltage range defined by **High Value** and the **Low Value**.

In the case of discrete control, the amount of reactive power supplied by this device changes in discrete amounts, thus the High Value should be greater than the low value. The necessary voltage range depends upon the size of the switched shunt blocks. In addition to a voltage range for discrete control, a specific **Target Voltage** can be specified as well. The target voltage will try to be met, either approximately under discrete control, or exactly under continuous control (either true continuous or discrete with a continuous shunt correction element.) The number of the regulated bus is shown in the **Reg. Bus #** field.

Generator Mvar Regulation

This option allows switched shunts to control generator Mvar output to better enable full var range inside the [inner power flow loop](#). During the first inner power flow loop, generator var limit checking is disabled for generators whose Mvar output is controlled by switched shunts. This will enable the use of the full reactive range of the switched shunts before generators hit their reactive limits. This corrects a problem in which the power flow fails to solve because generators are at their Mvar limits even though there is still reactive support available from switched shunts.

Switched Shunt Blocks

The amount of shunt reactive power (susceptance) is specified in the Switched Shunt Block field. The columns in this field correspond to different blocks of reactive power. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step (assuming 1.0 per unit voltage). You may model both capacitors and reactors. The reactors should be specified first, in the order in which they are switched in, followed by the capacitors, again in the order they are switched in. The sign convention is such that capacitors are positive and reactors negative. Shunt blocks are switched in order from left to right.

Control Parameters

Single Largest Step

This option only applies when a switched shunt is set on discrete control. If checked the switched shunt will switch in EITHER all of the available reactor blocks OR all of the capacitor blocks at once when the voltage falls outside the given range. Whether the reactor or capacitor blocks switch is determined by which limit is violated. A switched shunt with this option checked will only switch ONCE during a load flow solution, and then remains fixed at the new output for the remainder of the same solution calculation.

Allow switching in the inner power flow loop

This option is available for individual discrete shunts only. If this option is set, discrete shunts are treated as continuous in the [inner power flow loop](#). This means that they are treated as PV buses in the inner power flow loop. After the first inner power flow loop, the shunt nominal Mvar setting is rounded to positive infinity to the next discrete step. If any shunts exist that are allowed to switch in the inner power flow loop, the inner power flow loop is repeated again with the shunts being treated as discrete in the subsequent inner power flow loop. Shunts that are allowed to switch in the inner power flow loop will only switch if the global option to "Disable Treating Continuous SSs as PV Buses" is not checked.

Use Continuous Element

If this option is checked, then Simulator will use a continuous element to fine-tune a discrete controlled switched shunt by injecting or absorbing additional MVARs to try and obtain the target voltage of the controlled bus.

Minimum and Maximum Susceptance

The minimum and maximum susceptance range for the continuous correction element.

Use High Target Voltage

Check this box to use the target voltage specified in the **High Target Value** edit box when the regulated point goes above the High limit. This will give a different target value if the voltage goes out of range on the high end than the low end. If the voltage goes out of range on the low end, the original target value on the Parameters page will be used. If this box is unchecked, only the target value on the parameters page will be used, whether the violation is high or low.

Fault Parameters

Typically switched shunts are treated as open circuits in the zero sequence data for fault analysis. However, it is possible to define zero sequence admittance blocks to be used. The blocks work similarly to the load flow Switched Shunt Blocks discussed above. Usually there will be the same number of blocks in the zero sequence data as in the load flow data. Simulator will determine how many blocks were switched in for the power flow solution, and then use the zero sequence block data to calculate the zero sequence admittance for the same number of blocks.

Custom

This page of the dialog can be used to enter notes about the switched shunt. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be viewed and added from this page as well.

Switched Shunt Field Information

Switched shunt field objects are used primarily to indicate various quantities associated with switched shunt devices. Furthermore, some switched shunt field types, which are distinguished by an integrated spin button, may be used to change switched shunt device properties.

The Switched Shunt Fields Information Dialog can be used to modify the properties of individual switched shunt fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus to which the switched shunt associated with the field is connected. Use the dropdown box to view a list of all buses with switched shunts in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus to which the switched shunt associated with the field is connected. Use the dropdown box to view a list of all buses with switched shunts in the case with [valid area/zone/owner filters](#).

ID

ID of the switched shunt associated with the field.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

Switched shunt fields can be used not only to show various fields associated with switched shunt devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the switched shunt field. When the up spin button is clicked, the switched shunt field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the switched shunt field value is decreased by the same amount.

This field is only used for Switched Shunt Mvar fields. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the switched shunt field on the oneline.

Rotation Angle in Degrees

The angle at which the text is placed on the diagram, in degrees.

Anchored

If the *Anchored* checkbox is checked, the switched shunt field is [anchored](#) to its associated switched shunt, which means that it will move with the switched shunt.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of switched shunt field to show. The following choices are available:

Switched Shunt Mvar Total Mvar capacitance at the bus

Select a Field Choose from any of the different switched shunt fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

Zone Information (Edit Mode)

The Zone Dialog is used in the Edit Mode to view information about a zone and to move one or more buses from one zone to another. (See [Zone Information \(Run Mode\)](#) for help on the corresponding Run Mode version.) To view this dialog, open the [Zone Records Display](#) by clicking on **Aggregations > Zones** in the [Model Explorer](#). Then, right-click on the desired zone record and select **Show Dialog** to view this dialog.

The dialog has the following fields:

Zone Number

Zone number between 1 and 999. You can use the spin button immediately to the right of this field to move to either the next zone (click the up arrow) or the previous zone (click the down arrow).

Zone Name

Alphanumeric identifier for the zone. You can use this field to change the zone's name, provided you click either **Save** or **OK**.

Find By Number

To find a zone by its number, type the number in the **Number** field and click this button.

Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

Labels

To assign alternative identifying labels to the zone, click the Labels button, which will open the [Label Manager Dialog](#) listing all the labels assigned for the selected zone.

OK, Save, Cancel

OK saves any changes to the zone name and closes the dialog. **Save** saves any changes to the zone name, but does not close the dialog. **Cancel** closes the dialog ignoring any changes.

The zone dialog contains three additional pages of information:

Zone Buses Table

This table lists all of the buses in the zone. Number, name, voltage, area number, and area name are shown for each bus. This table can be used to move buses to a different zone. Select the bus or buses you would like to move with the mouse. Then, enter the **Destination Zone Number**, which is the zone to which you want to move the selected buses. You may enter a zone number that does not already exist, too, so that the buses will be moved to a brand new zone. In this case, be sure to provide the **Zone Name**, as well. Finally, click the **Move Selected Bus(es) to Destination Zone** button to implement the move.

OPF

The **Reserve Requirement Curves** options are only available with the [OPF Reserves add-on](#). More information about these options can be found in the [Area and Zone OPF Reserve Requirement Curves topic](#).

Custom

The Custom Page of the Zone Information dialog displays custom numbers or strings defined with the viewed zone. The [Memo](#) box is simply a location to log information about the zone. To log information about the zone, simply start typing your information or comments about the zone in the memo box.

Bus Information (Run Mode)

This dialog is used to view information about each bus in the system. It can be displayed by right-clicking on any bus on the oneline and choosing **Bus Information Dialog** or choosing **Show Dialog** from the [Bus Display](#). This dialog can only be reached in Run Mode, but has a similar [Edit Mode](#) counterpart. The Bus Information Dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus. You can use the small arrow immediately to the right of the number to view a list of all buses in the case with valid [display filters](#). Or you can use the spin button further to the right of the number to move to the next bus (click the up arrow) or the previous bus (click the down arrow).

Find By Number

To find a bus by its number, enter the number into the **Bus Number** field and then click this button.

Bus Name

Unique alphabetic identifier for the bus. You can use the small arrow immediately to the right of the bus name to view a list of all bus names in the case with [valid display filters](#).

Find By Name

To find a bus by its name, enter the bus name into the **Bus Name** field (case insensitive) and then click this button.

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Nom. Voltage

Nominal voltage of the bus.

Labels

Clicking on this button will open the [Label Manager dialog](#) listing all of the labels assigned for the selected bus.

Area Number, Name

Each bus is associated with an Area record. These fields show the number and name of this area. See [Area Records Display](#) for more details about areas.

Zone Number, Name

Each bus is associated with a Zone record. These fields show the number and name of the zone. See [Zone Records Display](#) for more details about zones. You can also use the [Zone Dialog](#) to list the buses in a particular zone and to easily move a group of buses from one zone to another.

Substation Number, Name

Each bus can be associated with a Substation record. By default, buses are not assigned to substations, and in that case these fields are blank. See the [Substation Records Display](#) topic for more details on adding substations.

Owner Number, Name

Each bus is associated with an Owner record. These fields show the number and name of the owner. See [Owner Records Display](#) for more information about Owners.

Voltage (per unit)

Bus voltage in per unit notation. You may enter a new per unit voltage magnitude. However, the only effect this has is changing the initial voltage guess used in the iterative solution. If you would like to change the reference voltage for a generator, please see [Generator Information Dialog](#).

Voltage (kV)

Bus voltage in actual kilovolts.

Angle (deg) and Angle (rad)

Voltage angle at the bus in degrees and radians. You may enter a new voltage angle. However, the only effect this has is changing the initial voltage guess used in the iterative solution EXCEPT AT THE SLACK BUS. Changing the angle for the slack bus will shift the voltage angle for all the buses in the slack bus' island by a similar amount.

Status

Status of the bus, either connected or disconnected. A disconnected bus is not energized. You can use this field to change the status of the bus. When the bus is initially connected, selecting **Disconnected** opens all of the transmission lines incident to the bus, disconnecting the bus from the rest of the system. Selecting **Connected** closes all of the lines incident to the bus unless they attach to another disconnected bus.

View Substation Dialog

Clicking on this button will open the [substation dialog](#) for the substation the bus is contained in.

View Owner Dialog

Clicking on this button will open the bus' [owner dialog](#).

Slack Bus

Checked only if the bus is a system slack bus. This value can only be changed in the Edit Mode.

View All Flows at Bus

Clicking on this button will open a [quick power flow list](#) for the current bus.

Device Info

Generator Information

Displays the total MW and Mvar generation at the bus. You cannot change either of these fields from this display. Select **View/Edit Generator Records** to view the individual generator records for the bus. Selecting this button displays the [Generator Dialog](#) for the first generator at the bus.

Load Information

Displays the total MW and Mvar load at the bus. You cannot change either of these fields from this display. Select **View/Edit Bus Load Records** to view the individual load records for the bus. Selecting this button displays the [Load Dialog](#).

Shunt Admittance

Shows the real and reactive components of the shunt admittance to ground. Entered in either MW or Mvar, assuming one per unit voltage. B is positive for a capacitor and negative for a reactor. If B corresponds to a switched device, consider using a [switched shunt](#).

Bus Voltage Regulation

This section lists the devices (if any) that are controlling the voltage at the bus.

Desired PU Voltage

If the bus is being regulated by one or more devices, this field will display the desired regulated voltage (in per unit) the devices are attempting to maintain.

Fault Analysis Load Parameters

This page of the display is only available for buses which have one or more load attached to the bus. The parameters on this tab are used when running a fault analysis study. The values represent the total load at the bus for the negative and zero sequence as equivalent admittances. By default, these values are zero. For load buses, these values can be changed by the user, or they can be specified by loading short circuit data from within the [Fault Analysis Dialog](#). It is also possible to define these values as non-zero at a bus where no load exists in the load flow, but it is not usually desirable to do so.

OPF

This tab is only available if you have the Optimal Power Flow (OPF) add-on tool for PowerWorld Simulator. This tab displays the MW marginal cost (Locational Marginal Price) for the bus when performing an OPF solution. The page also breaks down the LMP into its cost components.

Geography

The Geography page of the dialog provides information on the geographic location of the bus on the currently active oneline diagram.

Custom

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Substation Information (Run Mode)

This dialog is used in the Run Mode to view and modify information associated with a substation record. It displays different information from the [Edit Mode](#) version of the [substation dialog](#). To display it from [Run Mode](#), first select **Aggregation > Substations** in the [Model Explorer](#) to bring up the [Substation Records Display](#). Right-click on the substation of interest and choose **Show Dialog**. The Run Mode Substation Dialog has the following fields:

Substation Number

An integer identifier for the substation. You can use the spin button immediately to the right of this field to move to either the next substation (click the up arrow) or the previous substation (click the down arrow).

Substation Name and ID

Two alphanumeric identifiers for the substation.

Find By Number

To find a substation by its number, enter the number into the **Substation Number** field, then click this button.

Find By Name

To find a substation by its name, enter the name into the **Substation Name** field, then click this button.

Find By Sub ID

To find a substation by its substation ID, enter the ID into the **Substation ID** field, then click this button.

Find...

If the exact substation number, name and ID are not known, you can use the [Find Dialog](#) to search for and select a substation from a list of substations.

Labels

Clicking this button will open a dialog displaying the list of defined labels for the substation. New labels can also be added for the substation from the dialog as well.

View All Flows at Substation

Clicking this button will open a [quick power flow display](#) listing the buses contained in the substation.

Information

Load and Generation

Real and reactive load, generation, shunts, losses, and interchange for the substation.

Bus Voltages

Summary information on all buses in the substation, including total number of buses, number of dead (disconnected) buses, and minimum and maximum bus voltage and angle within the substation.

Available Gen MW/Mvar Ranges

Total amount of generation increase or decrease available for all generators in the substation.

Buses

The Buses table identifies the buses in the substation, and provides summary information on each.

Gens

The Gens table identifies the generators in the substation, and provides summary information on each.

Loads

The Loads table identifies the loads in the substation, and provides summary information on each.

Switched Shunts

The Switched Shunts table identifies the switched shunts in the substation, and provides summary information on each.

Tie Lines

The Tie Line Table identifies the flows on all of the substation's ties to other substation.

Custom

This page of the dialog can be used to enter notes about the substation. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be entered for storage with the substation object viewed.

Geography

Displays geographic information about the location of the substation, in Latitude and Longitude.

Generator Information (Run Mode)

This dialog is used to view information about each generator in the system. Many fields on this display can also be changed (except in Viewer). Here we describe the Run Mode version of the Generator Information Dialog. The [Edit Mode](#) version is very similar.

Bus Number

Unique number between 1 and 99,999 used to identify the bus to which the generator is attached. The dropdown list enumerates all generator buses in the case that meet the criteria established by [display filters](#). You may select a bus number directly from the dropdown list, or you may use the spin buttons to cycle through the list of generator buses.

Bus Name

Unique alphabetic identifier for the bus to which the generator is attached, consisting of up to eight characters. Use this dropdown box to view a list of all generator bus names in the case with [valid display filters](#).

ID

Two character alphanumeric ID used to distinguish multiple generators at a bus; '1' by default.

Find By Number

To find a generator by its number and ID, enter the number into the **Bus Number** field and the ID into the **ID** field. Then click the **Find By Number** button.

Find By Name

To find a bus by its name and ID, enter the bus name into the **Bus Name** field (case insensitive) and the ID into the **ID** field. Then click the **Find By Name** button.

Find...

If you do not know the exact generator bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the generator, either Closed (connected to terminal bus) or Open (not connected). You can use this field to change the status of the generator.

Area Name

Name of the area in which the generator's terminal bus is located.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected generator.

Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal bus' owner.

Fuel Type

Type of fuel used by the generator this model represents. In most cases, this field is unnecessary for normal load flow analysis, and hence the default value is Unknown. However, this value can be useful during the [Security Constrained OPF](#) analysis.

Unit Type

The type of unit the generator represents, such as combined cycle, steam, hydro, etc.

There additional sections of generator information available from the Run Mode generator dialog:

[Power and Voltage Control](#)

[Generator Cost Information](#)

[OPF](#)

[Fault Parameters](#)

[Owners, Area, Zone, Sub](#)

[Custom](#)

Generator Information: Power and Voltage Control

The **Power Control** grouping fields are used to show/change the values associated with the real power output of the generator.

MW Output

Current real power output of the generator.

Minimum and Maximum MW Output

Minimum and maximum real power output limits for the generator. Simulator will not let the MW output go below its minimum value or above its maximum value if the *Enforce MW Limits* option is exercised.

Available for AGC

Determines whether or not the generator is available for automatic generation control (AGC). Normally this box should be checked. However, there are times when you would like to control the generator output manually (such as if you are using the generator to remove a line limit violation), in which case you should leave this box unchecked. A generator is also placed on "manual" control any time you manually change its output. You could then place the generator back on AGC control by using this dialog.

Enforce MW Limits

If checked, the minimum and maximum MW limits are enforced for the generator, provided the **Enforce Generator MW Limits** field is also checked on the Limits Tab of the [Simulator Options Dialog](#). If this box is checked and a generator is violating a real power limit, the generator's MW output is immediately changed.

Participation Factor

The participation factor is used to determine how the real power output of the generator changes in response to demand when the generator is available for AGC and the area is on [participation factor control](#). When you open a case using the [PTI Raw Data Format](#), this field is initialized to the per unit MVA rating of the generator, since participation factor information is not stored in the PTI format.

Loss Sensitivity

Shows how the losses for an area will change for an incremental increase in the generation at the bus. This information is useful in determining the economic dispatch for the generation. The implicit assumption in calculating this field's value is that the incremental change in generation will be absorbed by the system "slack bus." This field cannot be changed.

The **Voltage Control** grouping is used to show/change values associated with controlling the voltage/reactive power output of the generator.

Mvar Output

Current reactive power output of the generator. You can manually change this value only if **Available for AVR** is not checked.

Min and Max Mvar Output

Specify the minimum and maximum allowable reactive power output of the generator.

Available for AVR

Designates whether or not the generator is available for automatic voltage regulation (AVR). When the AVR field is checked, the generator will automatically change its reactive power output to maintain the desired terminal voltage within the specified reactive power range. If a reactive limit is reached, the generator will no longer be able to maintain its voltage at the setpoint value, and its reactive power will then be held constant at the limit value.

Use Capability Curve

If checked, the generator's reactive power limits are specified using a reactive capability curve that prescribes the dependence of the generator's reactive power limits on its real power output. Otherwise, the fixed values given in the **Min Mvar Output** and **Max Mvar Output** fields are used. The generator reactive capability can be defined using the table that appears at the bottom of the dialog. Please see [Generator Reactive Power Capability Curve](#) for details.

Regulated Bus Number

Number of the bus whose voltage the generator is regulating. This is usually, but not always, the generator's terminal bus. Multiple generators can regulate the same remote bus, but the regulated bus must not be another generator bus. If the generator is at a slack bus, it must regulate its own terminal voltage. Select **Solution Details > Remotely Regulated Buses** from the [Model Explorer](#) to view the [Remotely Regulated Bus Records Dialog](#), which identifies all buses that are being remotely regulated.

Desired Reg. Bus Voltage

Specifies the desired per unit voltage for the generator at the regulated bus. The regulated bus need not be the terminal bus of the generator.

Actual Reg. Bus Voltage

Shows the actual per unit voltage at the regulated bus. If the generator is on AVR and has not reached a reactive power limit, the actual regulated bus voltage should be equal to the desired regulated bus voltage. This field cannot be changed.

Remote Reg %

This field is only used when a number of generators at different buses are regulating a remote bus (i.e., not their terminal buses). This field then specifies the percentage of the total reactive power required by the remote bus to maintain its voltage that should be supplied by this generator. The default value is 100.

Wind Control Mode and Power Factor

The Wind Control Mode effects how generator Mvar limits are treated and can be set to either *None*, *Boundary Power Factor* or *Constant Power Factor*.

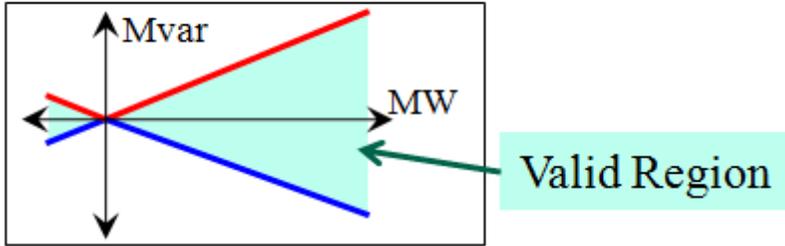
When the mode is *None*, then the generator behaves using the standard settings of the **Min and Max Mvar Output** and **Capability Curve** described above.

For both the *Boundary* and *Constant* modes, the Mvar limit magnitudes are determined from the actual **MW Output** and the **Wind Control Mode Power Factor** value.

$$\text{Magnitude} = \text{MWOutput} * \tan(\arccos(\text{Power Factor}))$$

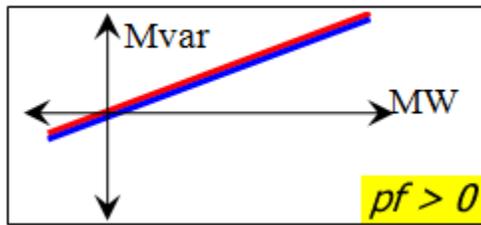
For *Boundary* mode, the **Max Mvar Output** is positive and the **Min Mvar Output** is negative. This provides a boundary under which the Mvar must operate.

Boundary Power Factor

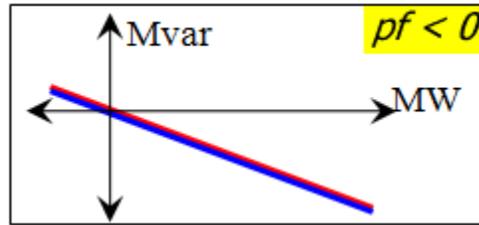


For *Constant* mode, **Max Mvar Output** and the **Min Mvar Output** are made the same with a positive **Wind Control Mode Power Factor** meaning the limits have the same sign as the actual **MW Output**, and a negative **Wind Control Mode Power Factor** meaning the limits are the opposite sign as the actual **MW Output**, thus the **Mvar output** is a function of **MW Output** and **Power Factor**.

Constant Power Factor



Constant Power Factor



Generator Options: Costs

The **Costs** tab of the Generator Information dialog (run mode) is used to show/change values associated with the cost of operation of the generator. See [Generator Cost Information](#) for details. Cost data can also be saved/loaded using the [Generator Cost Data files](#).

Output Cost Model

Cost Model

Simulator can model generators as not having a cost model, or having either a cubic cost model or a piecewise linear model. The cost model type you choose determines the content of the remainder of this dialog

Unit Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

Variable O&M

The Operations and Maintenance costs. Only used for cubic cost models.

Fixed Costs (costs at zero MW Output)

The fixed costs associated with operating the unit. These costs are independent of the generator's MW output level and is added to the cost prescribed by the cubic or piecewise linear model to obtain the total cost of operating the generator in \$/MWhr. The total fixed cost is the addition of the fuel cost independent value and the fuel cost dependent value multiplied by unit fuel cost.

Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form $C(P_{gi}) = (d * P_{gi}^3 + c * P_{gi}^2 + b * P_{gi}) * (\text{fuel cost}) + \text{fixed costs}$ (as described above), specify the cost curve's coefficients. The A coefficient is historically the fuel cost dependent fixed cost value, which is combined separately now with the fuel cost independent value. These coefficients can be specified only when you have chosen to use a cubic cost model.

Piecewise Linear Table

If you have chosen to use a piecewise linear cost model, a table appears that allows you to specify pairs of MW output levels and corresponding generator operating costs. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

Convert Cubic Cost to Linear

Use this option to create a piecewise linear cost function from the cubic cost function specified by the coefficients A, B, C, and D and the fuel cost. Specify the number of break points, and hence the number of segments, in the **Number of Break Points** field. Click the **Convert to Linear Cost** button to create the piecewise linear function that approximates the cubic cost function. This action switches **Cost Model** option to *Piecewise Linear* and displays the **Piecewise Linear Table** that identifies the piecewise linear curve's breakpoints.

Bid Scale / Shift

Cost Shift, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Marginal Cost (run mode only)

Shows the marginal cost of producing real power at the generator at its current output level, $dC_i(P_{gi})/dP_{gi}$.

ED/OPF Cost (run mode only)

This is the cost of production for this generator following an economic dispatch or optimal power flow solution, *including* the scaling from the cost shift and cost multiplier fields.

Unscaled Cost (run mode only)

The cost of production of the generator, *ignoring* the cost multiplier and cost shift. This cost is the result of the original cost function by itself.

OPF Reserve Bids

This tab is only available with the [OPF Reserves add-on](#). Detailed information about this tab can be found in the [Generator and Load OPF Reserves Bids](#) topic.

Generator Information: OPF

The fields on this tab display information regarding the generator's participation in an OPF load flow solution.

OPF MW Control

The type of control the generator is allowed during an OPF solution. The generator can be set to No control during OPF, control only if its AGC property is set to Yes, or to always be controlled by the OPF regardless of the AGC status of the generator.

Fast Start Generator

The generator is being treated as a fast start generator during the OPF solution.

Generator MW limits

The MW limits of a generator can be altered in this location if you wish for the generator to use different limits than originally assigned in the load flow case, without actually changing the original values. Simply change the Current Min MW Limit and Current Max MW Limit to alter the limits observed by the generator during and OPF solution.

MW Marginal Cost for Generator's Bus

The OPF solved marginal cost at the generator's terminal bus.

Initial, Final and Delta MW Output

The MW output information for the generator resulting from the OPF run.

Initial, Final and Delta Hourly Cost

The hourly cost information for the generator resulting from the OPF run.

OPF Results

This section shows the results of OPF for the generator.

Generator Options: Owners, Area, Zone, Sub

This information is located on the [Generator Information Dialog](#).

This tab is used to display or change the generator's owner information, area information, zone information, and substation information.

Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal buses' owner.

Owners

Currently, Simulator supports up to four owners for generators. To add an owner of a generator, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. To modify an owner's percentage of ownership, simply modify the value in the percentage field for that owner. If you set the percentage of an owner to 0, that owner will be removed from the list of owners for the device. You can also remove an owner from owning part of a device by changing the owner field for that owner to 0. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the generator dialog.

Area Number, Area Name

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The name and number of the substation to which the generator belongs, and it is the same than that of the terminal bus.

Load Information (Run Mode)

The Load Information Dialog can be used to inspect and modify the model of a bus load. To view the Load Information Dialog, select the load and choose **Show Dialog** from the [case information toolbar](#) or right-click on the load of interest and select **Load Information Dialog** from the resulting local menu. This is very similar to its [Edit Mode](#) counterpart. The dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the load is attached. The dropdown box provides a list of all load buses with [valid display filters](#). You can use the spin button to cycle through the list of load buses.

Bus Name

Unique alphabetic identifier for the bus to which the load is attached, consisting of up to eight characters. The dropdown box lists the names of all load buses in the case with [valid display filters](#).

ID

Two-character ID used to distinguish multiple loads at a bus. By default, the load id is equal to "1 ." An identifier of '99' is used to indicate an equivalent load.

Find By Number

To find a load by its number and ID, enter the number into the Bus Number field and the ID into the ID field. Then click this button.

Find By Name

To find a load by its name and ID, enter the bus name into the Bus Name field (case insensitive) and the ID into the ID field. Then click this button.

Find...

If you do not know the exact load bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the load, either Closed (connected to terminal bus) or Open (not connected). You can use this status field to change the load's status.

Labels

Clicking on this button will open the [Label Manager](#) dialog listing all the labels assigned for the selected load.

Area Number, Area Name

Number and name of the area the load is a member of.

Zone Number, Zone Name

Number and name of the zone the load is a member of.

Owner Number, Owner Name

Number and name of the owner the load is a member of. Loads DO NOT have to be owned by the same owner as the terminal bus.

Substation Number, Substation Name

Number and name of the substation the load is a member of.

Load Information

Base Load Model, Current Load

The Base Load Model fields are used to represent the amount of base real and reactive load at the bus. Usually this load is modeled as being "constant power," meaning that the amount of load is independent of the bus voltage magnitude. However, Simulator also permits modeling "constant current" load, for which the load varies in proportion to the bus voltage magnitude, and "constant impedance" load, for which the load varies in proportion to the square of the bus voltage magnitude. Values in these fields are specified in MW and MVR assuming one per unit voltage. All six fields in the Base Load Model section can be changed.

Load Multiplier

The actual load at the bus is equal to the base value multiplied by the corresponding load multiplier. The load multiplier is a value specifying how the load is scaled. The load multiplier depends upon the area load multiplier and the zone load multiplier. See [Load Modeling](#) for more details. The load multiplier value cannot be changed on this dialog.

Bus Voltage Magnitude

Voltage magnitude of the load's terminal bus.

Information about the remaining tabs on the dialog can be found at the following links:

[OPF Load Dispatch](#)

[Custom](#)

Line/Transformer Information (Run Mode)

The Line/Transformer Information dialog box is used to view information about each transmission line and transformer in the system. You may use this dialog also to change many of the properties of lines and transformers (except in Viewer).

The Run Mode version of this dialog is very similar in content to its [Edit Mode](#) counterpart.

The Line/Transformer dialog sports the following fields:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side.

To Bus Number and Name

To Bus number and name.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Find By Number

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

Find By Name

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

Find...

If you do not know the exact from and to bus numbers or names you are looking for, you can click this button to open the [advanced search engine](#).

From End Metered

This field is only used for lines and transformers that serve as tie lines, which are lines that join two areas. If this field is checked for a tie line, then the *from* end of the device is designated as the metered end. Otherwise the *to* end is metered. By default, the *from* end is metered. The location of the metered end is important in dealing with energy transactions because it determines which party must account for transmission losses.

From and To Bus Area Name

Names of the areas in which the From and To buses are located.

From and To Bus Nominal kV

From and To bus nominal voltage levels.

From and To Bus Voltage (p.u.)

The actual terminal bus voltages of the transmission element, in per unit.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected branch.

Parameters

Status

Current status of the device.

Per Unit Impedance Parameters

The resistance, reactance, the total charging susceptance (that is, B , not $B/2$), and the total shunt conductance of the device (in per unit).

Line Shunts

Select to view the [Line Shunt](#) Dialog. This dialog is used to only view the values of the line shunts in Run Mode. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer. If the line has shunts, the check box **Has Line Shunts** is checked.

MVA Limits

Ratings for the transmission line or transformer in MVA. Simulator allows the use of up to eight different limits.

Flows

These next fields show the actual real and reactive power flow at both ends of the device (because of real and reactive losses these numbers may be different), and its percentage MVA loading.

Transformer Info

This fields on this tab are enabled only if the branch is a transformer. See the [Transformer Modeling](#) help for more information on transformer types and controls.

Off-nominal Turns Ratio

The off-nominal tap ratio for the transformer.

Phase Shift Degrees

Phase angle for the transformer. This field is usually non-zero only for phase shifting transformers or wye-delta and delta-wye connected transformers, but can be non-zero for an LTC or fixed transformer on rare occasions.

Automatic Control Enabled

Check this box to enable automatic control of the transformer. Note that automatic control will be implemented only if (1) transformer control has been enabled for the transformer's area (see [Area Display](#) for details) and (2) transformer control has not been disabled for the entire case (via the *Disable Transformer Control* option on the *Power Flow Solution* tab of the [Simulator Options Dialog](#)). The Line/Transformer Dialog gives you convenient access to these control options through the *Case Control Enabled* and *Area Transformer Control Enabled* check boxes.

Automatic Control Type

Depending on the type of control of the transformer, the **Change Automatic Control Options** button will open up the control options for [LTC Control](#), [MVAR Control](#), or [Phase Shifter Control](#).

Area and Case Control Options

All these boxes must be checked for a transformer to be controlled. **Area Transformer Control Enabled** has to be checked for any transformer in that area to be controlled; **Case LTC Transformer Control Enabled** has to be checked for any transformer in the entire case to be controlled, and **Case Phase Shifter Control Enabled** has to be checked for any phase shifter in the entire case to be controlled. Checking these does NOT automatically make the entire area or entire case on control.

Series Capacitor

The parameters on this tab are enabled when the selected branch is a series capacitor. If the branch is a series capacitor, the **Is Series Capacitor** box will be checked. In addition, the **Status** field for series capacitors will be enabled, allowing you to change the Bypassed or In Service status of the series capacitor. The series capacitor status IS NOT the same as the branch status of Open or Closed.

OPF

The OPF tab is only visible if you have the OPF ([Optimal Power Flow](#)) add-on tool for PowerWorld Simulator.

Enforce Line Flow Limit

This check box must be checked if the branch limit is going to be enforced when running an OPF solution. If this box is not checked, the OPF routine will allow the branch to violate its branch limits.

Treat Limit as Equality Constraint

If checked, the OPF solution will attempt to solve the load flow while keeping the flow on the branch at its limit.

Maximum MVA Flow

The largest MVA flow value measured on the line, either at the From or To bus.

Present MVA Limit

The limit enforced by the OPF for the branch. This is set in the [OPF constraint options](#), and is related to the original branch limits.

Maximum Percentage

The highest percentage of flow measured on the line, either at the From or To bus.

Limit Marginal Cost

The cost of enforcing the branch MVA limit.

Included in LP

Specifies whether or not the branch flow and limit was included as a constraint in the OPF solution. In general, branches that are not near their limit and do not appear to be changing flow dramatically towards their limit will be ignored in the OPF calculation to speed up the solution. **No** and **Yes** indicate whether or not the OPF process determined that the line needed to be included. The user can initially force the branch to be included or not included with these two fields. By choosing **Always**, the branch will be included in the OPF solution constraints regardless of the propensity of the line to be approaching its limit.

MVA Flow Constraint Status

The constraint status for the line in the OPF solution will be shown here with the corresponding boxes checked by Simulator. These check boxes cannot be changed manually by the user.

From/To Bus MW Marginal Costs

Displays the marginal costs of the branches terminal buses, following the solution of the OPF.

Fault Info

The parameters on this tab are used when running a [fault analysis](#) study. The values represent the zero sequence impedance and zero sequence line shunt admittances for the analysis. By default, the

positive and negative sequence line impedances and line shunt admittances are the same as the load flow impedance. The same fields are used for transformers, along with the configuration field. The configuration field defines the winding type combinations for the transformer (wye, delta, etc.) As a default, Simulator assumes a grounded wye to grounded wye transformer, which has the same model as a transmission line. Usually transformers are not of this type, and the proper type would need to be defined either manually or loaded from an external file in order for the fault analysis to be accurate.

Owner, Area, Zone, Sub, PTDF

The **Default Owner (Same as From Bus)** read-only check-box indicates whether the line's owner is the same than the from bus' owner. Transmission elements can have up to four different owners, each with a certain owner percentage. To add an owner of a transmission element, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the line/transformer dialog.

The area, zone and substation to which the From and To buses belong, are also shown. If a PTDF calculation has been performed, the PTDF values for the viewed line will be displayed as well.

Custom

This page of the dialog can be used to enter notes about the transmission line. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. You can also enter custom strings and values that are also stored with the case.

Transformer AVR Dialog

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this display, click the *Automatic Control Options* button on the [Line/Transformer Information Dialog](#).

This dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control.

Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

Present Regulated Bus Voltage

The present voltage of the regulated bus.

Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the *Minimum Voltage* and *Maximum Voltage* fields), the *Voltage Error* field indicates by how much the voltage deviates from the control range.

Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

Present Tap Ratio

The tap ratio of the transformer for the current system state.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on *Power Flow Solution Tab* of the [Simulator Options dialog](#).

Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transformer Mvar Control Dialog

The Transformer Mvar Control dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers that are used to control the Mvar flow through the transformer. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#).

When used to control reactive power, the LTC transformer always controls the reactive power flow at the *from* end of the transformer (i.e., the tapped side), with positive flow assumed to be going through the transformer to the *to* bus. Therefore the regulated bus field is not used.

The dialog has the following fields:

Mvar Flow at *From* Bus

The current Mvar flow as measured at the *from* end of the line. This is the parameter the transformer tries to control.

Mvar Error

If the Mvar flow at the *from* end violates the limits defined by the *Minimum Mvar Flow* and *Maximum Mvar Flow* fields, the *Mvar Error* field indicates by how much the flow falls outside the control range.

Regulation Minimum Mvar Flow, Regulation Maximum Mvar Flow

Minimum and maximum allowable reactive power flow as measured at the *from bus*. The transformer attempts to regulate the reactive flow to fall within this range.

Regulation Target Type

As long as the regulated Mvar flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated Mvar flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated Mvar flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the Mvar flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated Mvar flow is above the maximum, and regulation minimum is used as the target value when the regulated Mvar flow is below the minimum.

Present Tap Ratio

The transformer's present off-nominal turns ratio.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Mvar to Tap Sensitivity

The amount of Mvar shift that would be implemented by switching one tap position from the current position. This sensitivity indicates the ability of the transformer to control Mvars.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transformer Phase Shifting Information

The Transformer Phase Shifting Dialog is used to view the control parameters of phase-shifting transformers. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#), provided that the *Phase Shift Control* option is chosen from the *Automatic Control* group.

Regulated Bus Number

Number of the terminal bus of the phase shifter regulated by the phase shifter. When control is active, the phase shifter will automatically change its phase shift to keep the MW flow at this bus at the desired value.

Current MW Flow

Current MW flow through the transformer measured at the regulated bus terminal.

MW Error

If the current MW flow falls outside the minimum/maximum MW flow limits, the *MW Error* field indicates by how much the flow violates the regulating range.

Regulation Minimum MW Flow, Regulation Maximum MW Flow

Minimum and maximum allowable MW flow through the phase shifter.

Regulation Target Type

As long as the regulated MW flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated MW flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated MW flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the MW flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated MW flow is above the maximum, and regulation minimum is used as the target value when the regulated MW flow is below the minimum.

Present Phase Angle (Degrees)

The phase angle of the transformer for the current solved system state.

Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

Step Size (Degrees)

Phase shift change per step in degrees.

MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an

existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

Transformer Impedance Correction Tables Dialog

The Transformer Correction Tables Dialog is used to view information about the transformer impedance correction tables. These tables are used on some LTC or phase shifting transformers to model the impedance of the transformer as a function of the off-nominal turns ratio or phase shift. The dialog has the following fields:

Transformer Impedance Correction Table Number

Number of the impedance correction table, between 1 and 63. Use the spin button immediately to the right of this field to step through the list of defined tables. If you have made changes to a particular table, you must click *Save* before moving to another correction table; otherwise, your changes will be lost.

Table Name

The name assigned to the table.

Table Entries

Used to insert/edit/delete the actual entries in the impedance correction table. In the first row, enter either an off-nominal turns ratio for an LTC transformer, or a phase shift in degrees for a phase shifting transformer. The entries in the first row must be entered in strictly ascending form. In the second row, enter the scale factor to apply to the transformer impedance. The transformer's nominal impedance is multiplied by the scale factor to obtain the actual value. Note that at least two columns must be used.

Right-click on the table to invoke its local menu, which allows you to delete and to insert columns. To insert a new column, click on the column before which you want to insert a new column and select **Insert New Point** from the local menu. To delete a column, position the cursor on the column you want to delete and select **Delete Point**.

Table is Used by the Following Transformers

Lists all the transformers in the case that use this impedance correction table. A single table may be used by any number of transformers. To associate a table with a transformer, use the [Transformer AVR Dialog](#) for LTC transformers or the [Transformer Phase Shifting Dialog](#) for phase shifters.

Switched Shunt Information (Run Mode)

The Switched Shunt Information Dialog box can be displayed by placing the cursor on its symbol and right-clicking. This is very similar to its [Edit Mode](#) counterpart. The dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the switched shunt is attached. This drop down list identifies the buses in the case with switched shunts that also have [valid display filters](#). Use the spin button to step through the list of shunts in the case. Note that only one switched shunt is allowed at each bus.

Find By Number

To find a switched shunt by its bus number, enter the number into the Bus Number field. Then click the **Find By Number** button.

Bus Name

Unique alphabetic identifier for the bus to which the switched shunt is attached, consisting of up to eight characters. This dropdown box lists the names of all the switched shunt buses in the case with [valid display filters](#).

Find By Name

To find a switched shunt by its name, enter the bus name into the Bus Name field (case insensitive). Then click the **Find By Name** button.

Shunt ID

Since multiple switched shunts are allowed on a single bus, each shunt is identified by a unique ShuntID.

Find...

If you do not know the exact switched shunt bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the switched shunt, either Closed (connected to terminal bus) or Open (not connected). On the oneline, the switched shunt can be opened by placing the cursor on the (red) circuit breaker box and clicking, and it can be closed by placing the cursor on the (green) box and again clicking. You can also use this status field to change the switched shunt's status. Note that the switched shunt is only available for automatic control when its status is closed.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected load.

Parameters

Nominal Mvar

Amount of reactive power that would be supplied by the switched shunt if its terminal voltage were one per unit (capacitive is positive).

Actual Mvar

Actual reactive power in Mvar being injected into the system by the switched shunt (capacitive is positive). The Actual Mvar field is equal to the Nominal Mvar field multiplied by the square of the terminal buses per unit voltage.

Nominal MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. In that case, it is possible for the bus shunt to have both a MW and MVAR component. The MW component will be displayed here. In general, switched shunts of other control types do not have MW components, and this field will not be displayed. If you change a switched shunt read as a bus shunt to another form of control, the MW component will remain, but has no controllability.

Actual MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. The value displayed is the actual real power in MW being injected into the system by the shunt. The Actual MW field is equal to the Nominal MVAR field multiplied by the square of the terminal buses per unit voltage.

Control Mode

Determines whether the switched shunt has a fixed value, or whether the amount of reactive power supplied by the device changes either in discrete steps or continuously in order to maintain its terminal voltage within the voltage range specified in the Voltage Regulation fields. This field can be changed (except in Viewer). However, for a switched shunt to be used for automatic control, three fields must be set correctly:

1. The *Control Mode* field must be set to either *Discrete* or *Continuous*
2. The corresponding area's *Auto Shunts* property must be true, and
3. The case-wide *Disable Switched Shunt Control* option, which can be set on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#), must not be checked.

Note: automatic control of switched shunts is disabled if the voltage regulation high value is not greater than the low value; they should not be equal unless in the continuous mode.

Note the additional control mode called Bus Shunt (Fixed). This is analogous to the shunt MW and MVAR values that can also be stored at the bus level. The difference is that bus shunts stored directly with the bus cannot be turned on and off in the load flow; rather they are always included in the load flow solution. Bus shunts that are represented as switched shunt objects on Bus Shunt control, however, are mathematically exactly the same and can be turned on or off. The reason for the differentiation of the Bus Shunt versus normal Fixed control is that the Bus Shunt control type is intended to identify the difference between a bus shunt and a transformer that MAY have controllability, but is currently turned off control by being set to a Fixed value.

Control Regulation Settings

Voltage Regulation

When the switched shunt is on automatic control, its reactive power is changed in discrete steps or continuously to keep the voltage at the regulated bus within the per unit voltage range defined by **High Value** and the **Low Value**.

In the case of discrete control, the amount of reactive power supplied by this device changes in discrete amounts, thus the High Value should be greater than the low value. The necessary voltage range depends upon the size of the switched shunt blocks. In addition to a voltage range for discrete control, a specific **Target Voltage** can be specified as well. The target voltage will try to be met, either approximately under discrete control, or exactly under continuous control (either true continuous or discrete with a continuous shunt correction element.) The number of the regulated bus is shown in the **Reg. Bus #** field.

Generator Mvar Regulation

This option allows switched shunts to control generator Mvar output to better enable full var range inside the [inner power flow loop](#). During the first inner power flow loop, generator var limit checking is disabled for generators whose Mvar output is controlled by switched shunts. This will enable the use of the full reactive range of the switched shunts before generators hit their reactive limits. This corrects a problem in which the power flow fails to solve because generators are at their Mvar limits even though there is still reactive support available from switched shunts.

Switched Shunt Blocks

The amount of shunt reactive power (susceptance) is specified in the Switched Shunt Block field. The columns in this field correspond to different blocks of reactive power. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step (assuming 1.0 per unit voltage). You may model both capacitors and reactors. The reactors should be specified first, in the order in which they are switched in, followed by the capacitors, again in the order they are switched in. The sign convention is such that capacitors are positive and reactors negative. Shunt blocks are switched in order from left to right.

Control Options

Single Largest Step

This option only applies when a switched shunt is set on discrete control. If checked the switched shunt will switch in EITHER all of the available reactor blocks OR all of the capacitor blocks at once when the voltage falls outside the given range. Whether the reactor or capacitor blocks switch is determined by which limit is violated. A switched shunt with this option checked will only switch ONCE during a load flow solution, and then remains fixed at the new output for the remainder of the same solution calculation.

Allow switching in the inner power flow loop

This option is available for individual discrete shunts only. If this option is set, discrete shunts are treated as continuous in the [inner power flow loop](#). This means that they are treated as PV buses in the inner power flow loop. After the first inner power flow loop, the shunt nominal Mvar setting is rounded to positive infinity to the next discrete step. If any shunts exist that are allowed to switch in the inner power flow loop, the inner power flow loop is repeated again with the shunts being treated as discrete in the subsequent inner power flow loop. Shunts that are allowed to switch in the inner power flow loop will only switch if the global option to "Disable Treating Continuous SSs as PV Buses" is not checked.

Use Continuous Element

If this option is checked, then Simulator will use a continuous element to fine-tune a discrete controlled switched shunt by injecting or absorbing additional MVARs to try and obtain the target voltage of the controlled bus.

Minimum and Maximum Susceptance

The minimum and maximum susceptance range for the continuous correction element.

Use High Target Voltage

Check this box to use the target voltage specified in the **High Target Value** edit box when the regulated point goes above the High limit. This will give a different target value if the voltage goes out of range on the high end than the low end. If the voltage goes out of range on the low end, the original target value on the Parameters page will be used. If this box is unchecked, only the target value on the parameters page will be used, whether the violation is high or low.

Fault Information

Typically switched shunts are treated as open circuits in the zero sequence data for fault analysis. However, it is possible to define zero sequence admittance blocks to be used. The blocks work similarly to the load flow Switched Shunt Blocks discussed above. Usually there will be the same number of blocks in the zero sequence data as in the load flow data. Simulator will determine how many blocks were switched in for the power flow solution, and then use the zero sequence block data to calculate the zero sequence admittance for the same number of steps and blocks.

Owner, Area, Zone, Sub

This tab is used to display or change the generator's owner information, area information, zone, and substation information

Area Number, Area Name

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

Owner Number, Owner Name

The owner number and name to which the switched shunt belongs. Note that you can change the owner of the switched shunt to be different than the owner of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different owner than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The substation number and name to which the switched shunt belongs. Note that you can change the substation of the switched shunt to be different than the substation of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different substation than that of the bus to which it is electrically connected.

Custom

This page of the dialog can be used to enter notes about the switched shunt. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be viewed and added from this page as well.

Zone Information (Run Mode)

This dialog is used in the Run Mode to view and modify information associated with a zone record. It displays different information from the [Edit Mode version of the zone dialog](#). To view this dialog, open the [Zone Records Display](#) by clicking on **Aggregations > Zones** in the [Model Explorer](#). Right-click on the zone of interest and choose **Show Dialog**. The Run Mode Zone Dialog has the following fields:

Zone Number, Zone Name

Number and name of the associated zone. Use either the combo box or the spin arrows to view the different zones.

Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

Labels

To assign alternative identifying labels to the zone, click the Labels button, which will open the [Label Manager Dialog](#) listing all the labels or aliases assigned for the selected zone.

The rest of the Run Mode Zone Dialog is divided into multiple pages:

Information**Load and Generation**

Real and reactive load, generation, shunts, losses, and interchange for the zone.

Generation AGC Range

These two fields show the total available MW reserve for generators in the zone that are on AGC and have nonzero participation factors. In other words, these fields show the total MW by which the generation in the zone can be increased or decreased using only generation that is presently on-line. The generator status, AGC status, and participation factor can be changed on the [Generator Records Display](#).

Tie Lines**Zone Tie Lines**

The Zone Tie Line Table identifies the flows on all of the zone's ties to other zones.

OPF**Average LMP for Zone**

The computed average locational marginal price of all buses contained in the zone.

LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the zone.

Minimum LMP

The minimum locational marginal price of all the buses in the zone.

Maximum LMP

The maximum locational marginal price of all the buses in the zone.

Reserve Requirement Curves

These options are only available with the [OPF Reserves add-on](#). More information about these options can be found in the [Area and Zone OPF Reserve Requirement Curves topic](#).

Zone Buses, Zone Gens, Zone Loads, Zone Switched Shunts

These pages show the case information display with the buses, generators, loads and switched shunts assigned to the zone.

Custom

The Custom Page of the Zone Information dialog displays custom numbers or strings defined with the viewed zone. The [Memo](#) box is simply a location to log information about the zone. To log information about the zone, simply start typing your information or comments about the zone in the memo box.

Labels

Using Labels for Identification

Most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) may have an alternative names assigned to them. These alternative names are called labels. Labels allow you to refer to equipment in the model in a way that may be unique to your organization. Labels may thus help clarify which elements are described by a particular set of data, especially when the short names employed by the power system model prove cryptic. Furthermore, since labels are likely to change less frequently than bus numbers, and since a label must, by definition, identify only one power system component, they may function as an immutable key for importing data from [auxiliary files](#) into different cases, even when bus numbering schemes change between the cases. Labels must be unique for devices of the same type, but the same label can be used for a device of a different type.

Information dialogs corresponding to buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces feature a button called **Labels**. If you press this button, the device's Label Manager Dialog will appear. The [Label Manager Dialog](#) lists the labels associated with the device and allows adding, deleting, and modifying labels. The **Labels (All)** field (variablename = LabelsAll) in a [Case Information Display](#) lists all of the labels that have been associated with a device. The device's primary label is the one that is listed first in this field. This field lists all labels assigned to a device as a comma-delimited string. Even though multiple labels may be assigned to a device, any label can be used to import data from [auxiliary data files](#). The Label Manager Dialog can be used to assign labels or the string of labels can be entered directly in the **Labels (All)** field.

Labels can be used to map data from an auxiliary data file to a power system device. Recall that [auxiliary data files](#) require you to include a device's key fields in each data record so that data may be mapped to the device. Labels provide an alternative key. Instead of supplying the bus number to identify a bus, for example, you can supply one of the bus's labels. The label will enable Simulator to associate the data with the device associated with that label. This mechanism performs most efficiently when the primary label is used, but other labels will also provide the mapping mechanism. The **Label (for use in input from AUX or Paste)** field (variablename = Label) is used for importing data using labels and is blank when viewing in a case information display. Keep in mind that all devices read via an auxiliary file using the label field should have a non-blank label. Otherwise, information for that device will not be read. Even if the primary or secondary key fields are provided with the device, as long as the label field is present, that is the only field that will be used to identify the device. New devices cannot be created by simply identifying them by label. Either the primary or secondary key fields must be present to create a new device and the label field should not be present.

Again, it is important to remember this: a single power system device may have multiple labels, but each label may be associated with only one device of a particular type. This is the key to enabling data to be imported from an auxiliary file using labels.

Saving Auxiliary Files Using Labels

All devices that can be identified by labels will have the **Labels (All)** and **Label (for use in input from AUX or Paste)** fields available in their case information displays. In order to save auxiliary files that identify devices by label, the two label fields should be added to the case information display prior to saving the data in an auxiliary file. Because the **Label (for use in input from AUX or Paste)** field will be blank when saved in the auxiliary file, this field must be populated with one of the labels in the **Labels (All)** field before loading the auxiliary file back in. Keep in mind that devices with blank labels cannot be identified when loading in an auxiliary file, so avoid saving auxiliary files by label if all devices do not have labels.

Many devices require [SUBDATA](#) sections. These sections have custom formats specific to the type of information that they contain. When saving auxiliary files with devices that require SUBDATA sections, the user can choose to use primary or secondary key fields or labels to identify devices in the SUBDATA sections. The user will either be prompted when saving the devices, or there is an option to change the key field to use when saving subdata sections on the PowerWorld Simulator Options dialog under the [Case Information Displays](#) category. When choosing to use labels, if a device has a label, it will be used. If it is a device that can be identified by buses and bus labels exist, bus labels will be used. Finally, if the device does not have a label and the buses do not have labels, the primary key for the device will be used for identification.

Devices that have SUBDATA sections that contain other devices that can be identified by labels include: contingencies, interfaces, injection groups, post power flow solution actions, and owners.

The setting to choose which identifier to use for the SUBDATA sections does not just apply to SUBDATA sections. Often when saving groups of options, this setting will apply to everything being saved with those options and not just the SUBDATA sections. This includes contingency options, ATC options, limit monitoring settings, and PVQV options. In these cases, there will be a prompt asking the user to decide which identifier to use in the auxiliary file.

Loading Auxiliary Files SUBDATA Sections Using Labels

The various SUBDATA sections that represent references to other objects can also be read using labels. Examples include contingencies, interfaces, injection groups, post power flow solution actions, and owners. When reading a SUBDATA section such as this, PowerWorld makes no assumption ahead of time about what identification was used to write this SUBDATA section. Instead, an order of precedence for the identification is as follows

	Identification	Explanation	Example
1	Key Fields	assumes that the strings represent Key Fields	BRANCH 8 9 1
2	Secondary Key Fields	assumes that the strings represent Secondary Key Fields	BRANCH Eight_138 Nine_230 1
3	Labels for component objects	the key/secondary key fields for some objects consist of references to other objects. An example of this is the BRANCH object that is described by the From Bus, To Bus, and Circuit ID. This assumes that labels of the component objects are used.	BRANCH Label8 Label9 1
4	Labels	Assumes that the string represents one of the Labels of the object	BRANCH LabelForBranch

Special Use of Labels

There are a few special cases where objects have fields that identify other devices. These devices can be identified by label but not in the conventional means because the label field applies to the object that contains the device and a SUBDATA section is not necessary. These special cases include: (Note all fields given below are by variable name because the use of labels is most relevant with auxiliary files.)

ATC Scenarios

[ATC Scenario](#) change records usually contain primary key fields to identify the devices that should be adjusted during the scenario. If using labels, these primary key fields will be replaced

with a single **Label** field. The use of this field is different because the Label field refers to the device in the change record and not to the change record itself. When labels are used with ATC scenarios, device labels only can be used. Bus labels cannot be used to identify devices for which no label exists but a bus label does.

ATC Scenarios are saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

ATC Extra Monitors

[ATC Extra Monitors](#) identify either branches or interfaces to monitor during the ATC analysis. These devices are identified in the **WhoAmI** field of ATC Extra Monitor records. Usually, the **WhoAmI** field is a special format that contains key field tags. Optionally, this field can use the label of the device for the extra monitor. If the device label is not available, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

ATC Extra Monitors area saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

Model Conditions

Devices in [Model Conditions](#) are usually identified by the **WhoAmI** field which is in a special format that contains key field tags. Optionally, this field can use the label of the device. If the device label does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

Model Expressions

[Model Expressions](#) contain Model Fields. Model Fields are identified by the **WhoAmI** fields in the Model Expressions. Usually, the WhoAmI fields are in a special format that contains key field tags. Optionally, these fields can use the label of the device associated with the Model Field. If the device does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

Bus Load Throw Over Records

[Bus Load Throw Over Records](#) are used with contingency analysis. These records have an option to identify the bus to which the load will be transferred by either number or name_kV combination. If choosing to identify objects by label, the **BusName_NomVolt:1** field will contain the label of the bus instead of the name_kV combination.

Bus Load Throw Over Records will be saved in an auxiliary file if choosing to [Save](#) settings on the Contingency Analysis dialog.

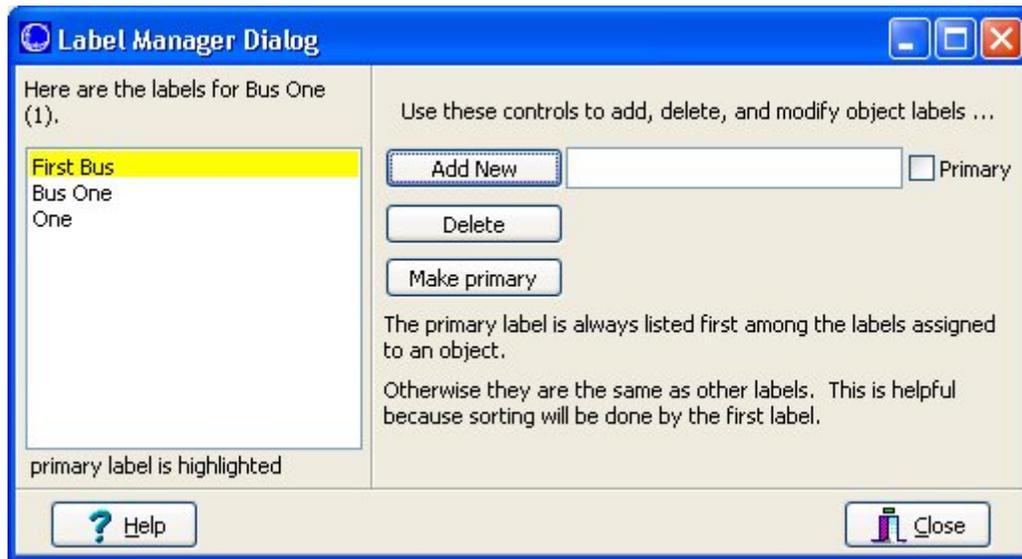
Injection Group Participation Points

All participation points and the injection groups to which they belong can be listed on the [Injection Group Display](#). Load, generator, and shunt devices that can be assigned to a participation point must be identified by bus and ID. The bus can be identified by either the number or name. When identifying by name, the **BusName_NomVolt** field is used to provide the name_kV combination for the bus. If choosing to identify devices by label, this field instead will contain the label of the device. If the device does not have a label but the bus does, the bus label will be used instead in conjunction with the ID of the device. Even if the device does contain a label, the ID field must be included in any auxiliary file that is going to be loaded because it is a key field. Injection groups can be included in other injection groups. Injection groups can be identified by label, even though this is not a normal thing to do. If any injection groups have labels and these injection groups are included in other injection groups, their labels will also appear in the BusName_NomVolt field. If they do not have labels, they will be identified by the injection group name that appears in the PPntID field.

Label Manager Dialog

The Label Manager Dialog allows the creation and modification of [labels](#) to identify power system devices. Labels can be used for most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) as alternate names for these devices. See the [Labels](#) topic for more information on how labels can be used.

The Label Manager Dialog can be accessed by clicking the Labels button on any of the information displays for objects that allow labels.



List of Labels

Multiple labels can be assigned to a single object. The list on the left of the dialog shows all of the labels that have been assigned with the primary label listed first and highlighted in yellow.

Add New

Click this button to add a new label after entering the new label in the box to the right of the button. If this new label should be the primary label, check the **Primary** box. Labels must be unique for each type of object. That means that there cannot be two buses labeled *One*. However, there could be objects of different types with the same label. There could be a bus and a generator both labeled *One*. If attempting to label an object with a label that has already been used by the same object type, the label will be assigned to the current object and removed from the object to which it was previously assigned.

Delete

Click this button to delete a label after first selecting the label in the list of labels.

Make Primary

Click this button to make a label the primary label after first selecting the label in the list of labels. The primary label will be moved to the top of the list. The primary label will always appear at the top of the list and will be used for sorting when sorting by label.

Area Information

The Area Information Dialog shows information about each area in the system. It displays an area's load, generation and losses; the area's scheduled interchange with other areas; options for controlling the area's generators, transformers, and shunts; the flows on its tie lines; and its operating cost information. You may view this dialog by doing any of the following:

- Select Areas from the [Model Explorer](#). This displays the [Area Display](#). Select the desired area and click the Show Dialog option on the [case information toolbar](#), or right-click on the record corresponding to the desired area to bring up the display's local menu and choose Show Dialog.
- Right-click in an empty portion of the oneline near a bus in the area of interest to display the [oneline's local menu](#). Select Area Information Dialog.
- While in Run Mode, right-click on an [Area Display Object](#) that represents an area and choose Information Dialog. Note: Doing this while in Edit Mode will show the [Area Display Options](#) dialog instead of the Area Information Dialog.

The Area Information Dialog contains the following information:

Number

A drop-down box that specifies the area number. Select an area from the drop-down box, or use the spin button to cycle through the list.

Find By Number

To find an area by its number, type the number in the **Number** field and click this button.

Name

A drop-down box that specifies the area's alphabetic identifier, which may be any length in Simulator. When saving area names to other load flow formats, the names are truncated at the maximum character length supported by that format.

Find By Name

To find an area by its name, type the name in the **Name** field and click this button.

Find...

If you do not know the exact area number or name you are looking for, you can click this button to open the [advanced search engine](#).

Super Area

To associate an area with a particular super area, either select an existing super area from the drop-down box, or type the name of a new super area.

Area Control Options

Select to change the area's method of [Automatic Generation Control \(AGC.\)](#)

Labels

To assign alternative identifying labels to the area, click the Labels button.

The rest of the Area Information Dialog is divided into nine pages of controls:

- [Info/Interchange](#)
- [Options](#)
- [Area MW Control Options](#)
- [OPF](#)
- [Tie Lines](#)

- [Buses](#)
- [Gens](#)
- [Loads](#)
- [Custom](#)

Area Information: Buses

The Area Buses grid shows a list of all buses that are in the area. The grid displays the bus number, bus name and other bus information. The Area Buses display is a [Case Information Display](#) and has options available to these types of displays.

Area Information: Gens

The Area Gens grid shows a list of all generators that are in the area. The grid displays the bus number, bus name, generator ID and other generator information. The Area Gens display is a [Case Information Display](#) and has options available to these types of displays.

Area Information: Loads

The Area Loads grid shows a list of all loads that are in the area. The grid displays the bus number, bus name, load ID and other load information. The Area Loads display is a [Case Information Display](#) and has options available to these types of displays.

Area Information: Area MW Control Options

Set Participation Factors...

Press this button to open the [Generator Participation Factors Dialog](#), which gives you control over how the participation factor for each generating unit is defined. If you decide not to prescribe participation factors using this button, the existing participation factors defined with each generator will be used. This button will be enabled only if the area has been set to Participation Factor area control.

Area Slack Bus Number

Identifies which bus to model as the area's slack bus.

Injection Group Area Slack

An additional area generation control option that can be utilized is the ability for Simulator to use a defined [injection group](#) as the sink for Area Slack control. When an injection group is selected, and the area is set on Area Slack control, any needed injection changes in the area will be accounted for by the devices in the injection group.

Some typical injection group options are also available here for handling the injection group dispatching during the Injection Group Area Slack Control. These options include:

Allow only AGC gen/load to vary:

Only generators and loads in the injection group that are designated as available for AGC control will be allowed to participate in injection changes.

Enforce generator MW limits (ignore case and area options):

Generators in the injection group can be designated to enforce their MW limits independent of the global case and area options for generator MW limit enforcement.

Do not allow negative loads:

If loads are decreasing demand as part of an injection group area dispatch, you can specify whether or not those loads should be allowed to go below 0 demand.

How should reactive power load change as real power load is ramped?

You can choose to keep the ratio of real and reactive power constant for each load that is included in the injection group, or you can specify a constant power factor that the MVAR value will be determined from when the MW value is changed.

Area Information: Info/Interchange

The Info/Interchange Tab serves as an accounting sheet for flows into and out of an area. It houses the following controls:

Load, Generation, Shunts, Losses, Interchange

These read-only fields express the total real and reactive load, generation, shunt compensation, losses, and interchange for the area.

Interchange

Interchange in MW between the area and all other areas (exporting power is positive). To prescribe an interchange with a specific area, simply enter the amount of power in MW to export to that area in the MW Export column. To specify an import, enter a negative value. If the recipient of the power is unknown, you may enter the net total in the **Unspecified MW Interchange** field found above the list of transactions. The net value of the MW imports and exports is displayed in **Transaction MW**.

Check the box **Only Show Areas with Nonzero Interchange** to display only those areas whose interchange is different from zero.

By default, one interchange record is initially defined between each area. The total interchange between two areas can be managed using this single record. However, it is possible to define multiple interchange records between the same areas. To do so, you can right-click in the interchange table and select Insert from the local menu. This will open the [Transaction Dialog](#), which will allow you to insert a new transaction. Transactions between the same two areas must have unique transaction IDs. Once you have multiple transactions defined between the same two areas, you can choose which transactions are enabled using the Enabled property in the Base Interchange table. This field is a toggleable field, meaning you can double-click in the field to toggle its value.

The total interchange defined for an area will be displayed in the summary field labeled **Interchange** in the area summary totals on the left side of the page.

AGC Tolerance

The MW tolerance is used in enforcing area interchange. When the absolute value of the ACE is less than this value, Simulator considers the area interchange constraint to be satisfied.

ACE (Area Control Error)

Current area control error (ACE) for the area in MW. Note that, when the constant frequency model is used, $ACE = \text{area generation} - \text{area load} - \text{area losses} - \text{scheduled area interchange} - \text{area shunt MW}$.

Area Has Multiple Islands

This is an informational field that cannot be changed. If checked, then Simulator has detected that the area has devices in separate [electrical islands](#) in the system. This is important because it is sometimes impossible for an area using some form of generation control ([AGC](#)) to solve for the generation dispatch in the entire area ($\text{Area Generation} = \text{Area Load} + \text{Area Losses} + \text{Area Interchange} + \text{Area Shunt MW}$) when the area is not entirely within the same electrical island. If Simulator is unable to solve for areas that span multiple islands, the AGC status of the area will automatically be set to "Off AGC" and a warning message will be written to the Simulator [message log](#).

Area Information: Options

Report Limit Violations

If checked, limit violations for this area are reported. Limits violations are reported in the [Limit Violations List](#).

Generation AGC Values

These fields indicate the total available AGC range for all the generators in the area **that are specified as being on AGC and have nonzero participation factors**. That is, these fields show the total amount the generation in the area that can be increased or decreased using the presently on-line generation. This may be referred to as the *spinning reserve*. The generator status, AGC status, and participation factor can be changed on the [Generator Records Display](#).

Load MW Multiplier Value

This value is used for scaling the area MW load. The base load remains unchanged.

Load MVAR Multiplier

This value is used for scaling the area MVAR load. The base load remains unchanged.

Automatic Control Options

The Automatic Control Options section provides a convenient mechanism to enable or disable automatic control of switched shunts and transformers in the area.

This section also provides a check box for setting if the generators in the area should enforce their Generator MW Limits or not. Typically this should be checked. Note that even if unchecked, the [OPF](#) routine will ignore this setting and will always enforce generator MW limits during an OPF solution.

Include Loss Penalty Factors in ED

If this box is unchecked, then the economic dispatch for the area is calculated assuming that the area is lossless. Otherwise, the solution will incorporate losses when computing the economic dispatch. The penalty factors gauge the sensitivity of the area's losses to changing injection at specific generators. The option to calculate loss penalty factors is relevant only when the area operates according to *Economic Dispatch Control*. Usually, if the system's cost curves are relatively flat, the inclusion of losses in the solution will not have much of an effect on the dispatch.

Economic Dispatch Lambda

The lambda value calculated during the economic dispatch computation. This field is valid only when the area is on economic dispatch control.

Area Information: Tie Lines

The Tie Lines display shows the flow on all of the tie lines for the selected area. This display is a [case information display](#) and contains many of the options available to these types of displays.

The default fields that are shown in the grid identify each line and show the flow on each line.

Near Area Name, Near Area Number

These fields identify the area name and number of the tie line terminal located in the selected area.

Near Number, Near Name

These fields identify the bus number and name of the tie line terminal located in the selected area.

Far Area Name, Far Area Number

These fields identify the area name and number of the tie line terminal located in the other area.

Far Number, Far Name

These fields identify the bus number and name of the tie line terminal located in the other area.

Ckt

This is the circuit identifier for the tie line.

Meter MW, Meter Mvar

These are the real and reactive power **flowing on the tie line from the selected area to the other area.** (Real and reactive power flowing on line from the Near bus to the Far bus.)

Status

This field gives the status of the tie line as *Open* or *Closed*.

MW Loss, Mvar Loss

These fields show the real and reactive power losses on the tie line.

Area Information: OPF

The OPF Tab contains information regarding the [Optimal Power Flow \(OPF\)](#) solution data for an area. It houses the following information:

Average LMP for Area

The computed average locational marginal price of all buses contained in the area.

LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the area.

Min/Max LMP

The minimum and maximum locational marginal price of all the buses in the area.

Total Generator Production Cost (Scaled)

The scaled cost includes the Cost Shift and Cost Multiplier. These two values can be defined for each generator, and allow the user a way to assess changes to the LMP results when a generator's cost or "bid" is modified, without actually changing the original generator cost or bid curve. The scaled cost function for each generator is equal to:

$(\text{original cost function} + \text{cost shift}) * \text{cost multiplier}$

Total Generator Unscaled Production Cost

The total unscaled generator production cost, based on the original generator cost or bid curves.

Total Generator LMP Profit

The profit of the generators in the area based on the Locational Marginal Prices (LMPs) determined by the OPF solution. The profit is determined as:

$\text{LMP Price} * \text{MW Output} - \text{Unscaled Cost Function}$

Cost of Energy, Loss, and Congestion Reference

Specify a reference for determining the cost of energy, loss and congestion. The choices are Existing loss sensitivities directly, Area's Bus' Loads, [Injection Group](#), or a specific bus.

Reserve Requirement Curves

These options are only available with the [OPF Reserves add-on](#). More information about these options can be found in the [Area and Zone OPF Reserve Requirement Curves topic](#).

Area Information: Custom

The Custom page of the [Area Information](#) dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for areas. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the Area Information dialog is simply a location to log information about the area. To log information about the area, simply switch to the Custom page on the dialog, and start typing your information or comments about the area in the memo box.

Area Field Options

Area field objects are used to show different values associated with areas and the system. This dialog is used to view and modify the parameters associated with these fields.

Area Number

Area number associated with the field. When you insert fields graphically, this field is automatically set to the area number associated with the closest bus on the oneline. With most types of area fields, an *Area Number* of 0 is valid and defines the field as showing values for the entire system.

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Fields

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Other Area Number

Some of the fields, such as **MW Flow to Other Area**, require that a second area be specified. If applicable, enter the second (other) area here.

Other Area Transaction ID

Since it is now possible to have more than one base transaction defined between the same two areas, base transactions must now also have a unique ID to distinguish between transactions among the same two areas. If you are displaying a field that pertains to the display of the scheduled flow between areas, the ID of the transaction in question will also need to be entered here.

Delta MW per Mouse Click

This value is used only with the **Sched Flow to Other Area** field type. When there is a nonzero entry in this field, and the field type is **Sched Flow to Other Area**, a spin button is shown to the right of the area field. When the up spin button is clicked, the flow to the other area is increased by this number of MW; when the down button is clicked, the scheduled flow is decreased by this amount.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Field Value

Shows the current output for the area field. Whenever you change the **Type of Field** selection, this field is updated.

For the **Sched Flow to Other Area** field type only, you can specify a new value in MW. Exports are assumed to be positive.

Anchored

If checked, the area field will be anchored to a corresponding [Area Object](#). If the area object is moved on the diagram, the text field will move with it.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of bus field to show. The following choices are available:

Name Area name (eight characters maximum). The *Area Number* field must correspond to a valid area.

Number Area number (1 - 999). The *Area Number* field must correspond to a valid area.

MW Load, Mvar Load If the area number is nonzero, then these fields show Total MW or Mvar load for the area. If the area number is zero, these fields show the total load in the entire system.

MW Generation, Mvar Generation If the *Area Number* is nonzero, then these fields show Total MW or Mvar generation for the area. If the *Area Number* is zero, these fields show the total generation in the entire system.

MW Losses, Mvar Losses If the *Area Number* is nonzero, then these fields show Total MW or Mvar losses for the area. If the *Area Number* is zero, these fields show the total losses in the entire system.

ACE (MW) Area Control Error in MW for the area. The *Area Number* field must correspond to a valid area.

Hourly Cost (\$/hr) If the *Area Number* is nonzero, then this field shows the hourly cost for the area. If the *Area Number* is zero, these fields show the hourly cost for the entire system.

Total Cost (\$) If the *Area Number* is nonzero, this field shows the total cost incurred by the area since the beginning of the simulation. If the *Area Number* is zero, this field shows the total cost incurred throughout the system since the beginning of the simulation.

MW Flow to Other Area,

Mvar Flow to Other Area Total MW or Mvar flow from the area specified in the *Area Number* field to the area specified in the *Other Area Number* field. The *Area Number* field must correspond to a valid area. If the *Other Area Number* field is zero, this field shows the area's total MW or Mvar exports.

Sched. Flow to Other Area Scheduled MW transaction from the area specified in the *Area Number* field to the area specified in the *Other Area Number* field, and with the Transaction ID given in the *Other Area Transaction ID* field. The *Area Number* field must correspond to a valid area. If the *Other Area Number* field is zero, this field shows the area's total scheduled MW transactions. If the *Delta per Mouse Click* value is nonzero, you can use the spin arrows on the oneline to change this value. Also, you may directly enter a new value in the *Field Value* field.

Load Schedule Multiplier (MW Only) Indicates the current value of the MW multiplier applied to the area's loads.

AGC Status Displays the AGC status of the area.

Select a Field Choose from all possible information fields that can be displayed for areas.

Select **OK** to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

Super Area Information Dialog

The Super Area Information Dialog displays information pertaining to [Super Area Display](#). It summarizes the super area's real and reactive load and generation, scheduled transactions, and constituent areas. It also allows you to manage the list of areas to include in the super area and to designate how its generation should be controlled. To display the Super Area Information Dialog, right-click on the [Super Area Records Display](#) and select **Show Dialog** from the resulting local menu.

The Super Area Information Dialog has the following fields:

Area in Super Area

Name

The list of all super areas that have been defined in the case are listed in this dropdown box. Select one of the super areas to display its information, or type a new name in the box and click **Add New** to define a new super area.

Rename

Click this button to change the name of the super area specified in the Name field.

Save

Saves the super area of the name specified in the list of super areas.

Delete

Click this button to delete the super area specified in the Name field.

Area in Super Area

Areas in Super Area

Lists the areas contained in the super area. Right-click on a row and select **Remove** to remove the area from the super area.

Super Area Control Options

Specify the type of generation control to employ for the super area. The super area may be removed from area control or employ Participation Factor or Economic Dispatch control. See [Area Control](#) for details on these types of generation control.

An additional type of control available to Super Areas is to **Use Area Participation Factors**. If the super area is set to participation factor control, and this option is NOT checked, then the generators within the super area respond by redispatching to meet the entire super area generation change based on their own participation factors.

However, if this option IS checked, then an additional level of participation control complexity is added for the super area. First, the total super area generation change is divided across each area forming the super area, according to the participation factors of the AREAS, as set in the grid on this dialog. Once the total generation change has been determined for each area within the super area, then the generators within each individual area are dispatched using [area participation factor control](#) to meet each specific area's determined generation change.

Use Area Participation Factors

If the super area control is set to Participation Factor control, you can specify participation factors for each area in the list of areas that indicate how each area should contribute towards changes in generation for the super area. For example, if you have three areas in the super area and you set the participation factors to 1, 1 and 2, the three areas will contribute to the total change in

generation in the super area by ratios of 25%, 25%, and 50%, respectively. The contribution of each area is then further divided among the generators within each area by the participation factors of the individual generators.

This option will be applied when calculating [TLR](#), [PTDF](#), [voltage](#), and [ATC](#) sensitivities. When calculating these sensitivities, the assumption is that the super area is on participation factor control even if the control method is something other than this. If this option is selected, it will be enforced regardless of the control method when calculating these sensitivities.

New Area Name

Use the dropdown box to select an area to add to the super area. Click the **Add New Area by Name** to add the selected area to the super area.

New Area #'s

Enter in a list of area numbers separated with dashes or commas and click **Add New Areas by Number** button to add the areas to the Super Area

Summary Information

ACE Total (MW)

The total area control error for the super area. It is calculated by summing the ACE for all areas comprising the super area.

ACE Tolerance (MW)

The Area Control Error tolerance observed for the super area during a load flow solution.

Total Scheduled Transactions

Lists the total scheduled import or export for the super area. It is computed by summing the scheduled interchange for all areas comprising the super area. Exports are positive.

Lambda

The marginal cost associated with the super area. Lambda is valid only for super areas that are on economic dispatch control.

Hourly Cost

Current average hourly cost for the super area in \$/hour.

Load, Generation, Shunts, Losses, Interchange

The Load and Generation section of the Super Area Information Dialog accounts for the real and reactive power flows into, out of, and within the super area. Each quantity is computed by summing over the areas comprising the super area.

OPF

The OPF Tab contains information regarding the [Optimal Power Flow \(OPF\)](#) solution data for a super area. It houses the following information:

Average LMP for Area

The computed average locational marginal price of all buses contained in the area.

LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the area.

Min/Max LMP

The minimum and maximum locational marginal price of all the buses in the area.

Total Generator Production Cost (Scaled)

The scaled cost includes the Cost Shift and Cost Multiplier. These two values can be defined for each generator, and allow the user a way to assess changes to the LMP results when a generator's cost or "bid" is modified, without actually changing the original generator cost or bid curve. The scaled cost function for each generator is equal to:

$(\text{original cost function} + \text{cost shift}) * \text{cost multiplier}$

Total Generator Unscaled Production Cost

The total unscaled generator production cost, based on the original generator cost or bid curves.

Total Generator LMP Profit

The profit of the generators in the area based on the Locational Marginal Prices (LMPs) determined by the OPF solution. The profit is determined as:

$\text{LMP Price} * \text{MW Output} - \text{Unscaled Cost Function}$

Cost of Energy, Loss, and Congestion Reference

Specify a reference for determining the cost of energy, loss and congestion. The choices are Existing loss sensitivities directly, Area's Bus' Loads, [Injection Group](#), or a specific bus.

Custom

This page of the dialog can be used to enter notes about the super area. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be entered on this page in the form of floating point, integer, and string fields. These custom fields are stored with the super area and can be used for various tasks in Simulator.

Super Area Field Information

Super area field objects are used to show different values associated with super areas. This dialog is used to view and modify the parameters associated with these fields. Note that the super area name itself cannot be changed on this dialog.

Super Area Name

Select the name of the super area for which you are inserting or viewing information of a super area field.

Find...

If you do not know the exact super area name you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Anchored

Toggles if the field should be anchored to an associated super area object on a diagram.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of super area field to show. The following choices are available:

Name Super area name (eight characters maximum). The super area name field must correspond to a valid super area.

MW Load, Mvar Load Total MW or Mvar load for the super area.

MW Generation, Mvar Generation Total MW or Mvar generation for the super area.

MW Losses, Mvar Losses Total MW or Mvar losses for the super area.

ACE (MW) Area Control Error in MW for the super area. The super area name field must correspond to a valid super area.

MW Exports Total MW exports for the super area.

Hourly Cost (\$/hr) The hourly cost for the super area.

AGC Status Displays the AGC status of the super area.

MW Marginal Cost (\$ / MWhr) MW marginal cost for the super area.

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

Interface Information

The Interface Dialog is used to create, modify, or delete interface records in both the Edit and Run Modes. This dialog has the following controls:

Interface Name

An alphanumeric identifier for the interface of up to 24 characters. Use the dropdown box or the spin button to navigate through the list of existing interface records.

Interface Number

A numeric identifier for the interface, between 1 and 32,767.

Find Interface...

If you do not know the exact interface name or number you are looking for, you can click this button to open the [advanced search engine](#).

Add New Interface

Click the Add New Interface button to define a new interface from the Interface Dialog. When you click this button, the **Interface Name** field and **Interface Elements** table are cleared, requiring you to enter a new name and new elements.

Delete Interface

Select this button to delete the currently displayed interface. Once the interface record has been deleted, the **Interface Name** field displays the previous interface record, if any. If there are not previously defined interface records, Simulator will close the dialog.

Interface Limits

Specify the possible limits for net interface flow. As for transmission lines and transformers, up to eight distinct limits can be specified for interfaces. Which limit set is used can be controlled from the *Limits Tab* of the [Simulator Options Dialog](#) or the [Line and Transformer Limit Violations Display](#).

Monitoring Direction

This area allows the user to change the flow monitoring direction. Clicking on the interface in the **Interface Elements** area will bring up the **Interface Elements Dialog**. This is where the default direction is set. To keep this default direction, choose FROM → TO, to change this direction, choose TO → FROM, and to monitor both directions, choose Both Directions.

Noncontingent MW Flow Contribution

Indicates the present net flow through the interface elements, not including affects of contingencies, in MW.

Contingent MW Flow Contribution

The amount of flow that would be added to the interface flow if the contingency defined with the interface were to occur, in MW.

Total MW Flow

The sum of the non-contingent MW flow and Contingent MW flow, for the total MW flow considered on the interface, in MW.

PTDF Value (%)

The [Power Transfer Distribution Factor](#) for the interface, if calculated.

Interface Elements

The Interface Elements Table lists each element comprising the interface. If Simulator is in Run Mode, the table will also show the present flow through each element. To edit or delete an existing element in the table, click on it to bring up the [Interface Element Dialog](#). Use the Interface Element Dialog to modify or delete the element.

Insert New Element

Click the Insert New Element Dialog to add a new element to the interface using the [Interface Element Dialog](#).

Click **OK** to save any changes you have made and to close the Interface Information Dialog. Click **Save** to save your changes but to leave the dialog open so that you can view and modify other interface records. Click **Cancel** to close the dialog without saving your latest change.

Element Identifiers

Choose whether to show the interface descriptions using the interface names, the interface numbers, or a combination of both.

OPF

The OPF page will only be visible if you have the Optimal Power Flow add-on for PowerWorld Simulator. This page contains interface information relating to performing an OPF solution.

Enforce Interface Flow Limit

If checked, the Interface limit will be checked for enforcement during an OPF solution.

Treat Limit as Equality Constraint

If checked, the OPF routine will try and maintain the flow on the interface at its limit value. Otherwise the limit will be treated as the maximum for an inequality constraint.

MW Flow on Interface

The actual MW flow on the interface.

Present MW Limit

The currently defined limit for the interface.

Percentage of Limit

The MW flow on the interface as a percentage of the limit.

Limit Marginal Cost

The incremental cost of maintaining the flow on the Interface at its limit value.

MW Flow Constraint

The constraint status of the interface in the OPF solution is shown here with the corresponding boxes checked by Simulator. These boxes can not be changed by the user.

Interface Element Information

The Interface Element Dialog is used to redefine or to add the individual elements comprising an [interface](#). Individual lines or transformers, inter-area ties, inter-zone ties, line contingencies, DC lines, injection groups, generators, and loads may make up an interface. The Interface Element Dialog allows you to add all three varieties of interface elements to an interface.

The Interface Element Dialog comes in two very similar forms, depending upon how it was invoked. The dialog may be called from the [Interface Information Dialog](#) by clicking on either the Interface Elements Table or the Insert New Element button.

The Interface Element Dialog contains the following controls:

Element Type

Specifies the type of interface element being investigated or added. Interface elements can be any line or transformer or, more specifically, they can be groups of lines and transformers that tie two areas or two zones together. Interfaces can also contain line contingencies, DC lines, generators, loads, and injection groups.

When the selection for Element Type changes, the available **Element Identifiers** change to allow you to pick the appropriate elements of that type.

Element Identifiers

Depending upon the **Element Type** selection, different element identifiers are required to designate the element to add to the interface. You can search through the list of identifiers to find the particular elements you wish to include. Note that the flow direction on transmission line, transformer, and DC line elements will be dependent on which end of the line you choose as the near bus. The flow will always be measured in the direction of near bus to far bus. The flow will be positive when flowing from near to far, and negative when flow from far to near. In addition, line elements also have an additional setting labeled **Monitor Flow at To End**. This determines which magnitude of flow should be reported. If checked, the flow at the To end of the line will be reported, otherwise the flow magnitude at the From end is used.

Insert

If you came to the Interface Element Dialog by pressing the Insert New Element button on the Interface Information Dialog, only the **Insert**, **Cancel**, and **Help** buttons will be available. Click the Insert button to add the element you have just defined to the list of elements comprising the interface. After you click Insert, the dialog will disappear, and the Interface Elements table on the Interface Information Dialog will contain the element you just added.

Replace, Delete

If you arrived at the Interface Element Dialog by clicking on an element in the Interface Elements Table of the Interface Information Dialog, the **Replace**, **Delete**, **Cancel**, and **Help** buttons will be visible. Click **Replace** to modify the interface element according to your specifications on this dialog. Click **Delete** to remove the element from the interface definition.

Cancel

Click **Cancel** to close the Interface Elements Dialog without saving your changes.

Interface Field Information Dialog

Interface field objects are used to show the different values associated with [interface records](#). This dialog is used to view and modify the parameters associated with these fields. The dialog has the following fields:

Interface Name

Case insensitive name of an existing interface (12 characters maximum).

Find...

If you do not know the exact interface name you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The rotation angle at which the text field should be displayed.

Anchored

If checked, the interface field is anchored to its associated interface display object.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Type of field to show for the interface record. Interface fields may show the interface's name, its MW Limit, MW Flow, percent loading, or any other interface specific field available by clicking the Select a field option and choosing a field from the drop-down list.

Interface Pie Chart Information Dialog

Interface Pie Chart objects are used to graphically show the percentage flow associated with [interface records](#). This dialog is used to view and modify the parameters associated with these fields. The dialog has the following fields:

Interface Name

Case-insensitive name of an existing interface (12 characters maximum).

Size

Size of the pie chart. Note that the pie chart's size and color can be set to change automatically when the interface's loading is above a specified limit. Please see [Online Display Options](#) for details.

Actual Value (read-only)

In Run Mode, this field shows the current MW loading for the interface.

Percent (read-only)

In Run Mode, this field shows the current percentage loading for the interface; if the MVA rating is zero, the percentage is defined as zero, as well.

Anchored

If checked, the interface pie chart object is [anchored](#) to its associated interface.

MW Rating

MW limit for the interface, using the current limit set for the case.

Style

The style of the pie chart. It can be set individually for the particular pie chart, or it can use the style specified in the [Interfaces Pie Chart Options](#).

Automatically Inserting Interfaces in Case

The **Inserting Interfaces Dialog** is used to insert a group of [interfaces](#). Only interfaces between adjacent areas or adjacent zones can be inserted automatically; single-branch interfaces between buses must be inserted with line insertion options. Adjacent areas or zones are those that share at least one tie line. To reach this dialog, first open the Interfaces Display by selecting **Aggregations > Interfaces** from the [Model Explorer](#), and then from the [case information toolbar](#) select **Records > Auto Insert Interfaces**. This dialog is NOT brought up through the menu item **Auto Insert > Interfaces** found on the [Quick Insert](#) ribbon group of the [Draw](#) ribbon tab, which is for inserting [interface objects](#) on an oneline diagram.

The Inserting Interfaces Dialog sports the following fields:

Type of Interfaces to Insert

Select the type of interfaces to insert. Area-to-area interfaces join adjacent areas, while zone-to-zone interfaces join adjacent zones. The name of the new interface defaults to "Area1- Area2" or "Zone1-Zone2" with an **Optional Prefix**.

Optional Prefix

This field allows you to specify an optional prefix of up to three characters. Use this prefix to avoid duplicating names, particularly when some of the areas or zones have the same name.

Delete Existing Interfaces

If this option is checked, then all existing interfaces are deleted before inserting the new interfaces. By default, this option is checked. If this option is not checked, the existing interfaces are not deleted. However, new interfaces will automatically overwrite any existing interfaces having the same name.

Only Insert Between Areas/Zones with Area/Zone Filters Set

If this option is checked, the set of potential areas or zones for inserting interfaces is limited to those for which the area/zone filter setting is *Yes*.

Limits

Simulator can either calculate an interface rating based on the ratings of the components included in the interface, or the user can specify a set of ratings to be used for the interface. If neither of these options is used to set an interface limit, then by default, the interface limits are left as 0, indicating no limit has been applied.

Insert Interfaces

Click this button to insert the interface records into the case.

Cancel

Closes the dialog without modifying the list of interfaces.

Nomogram Information Dialog

The Nomogram Dialog is used to create, modify, or delete nomogram records in both the Edit and Run Modes. Nomograms are used for combining a pair of [interface objects](#) for the purpose of monitoring a combined flow restriction on the two interfaces together.

This dialog has the following controls:

Nomogram Name

An alphanumeric identifier for the nomogram.

Interface A

The first interface forming the interface pair. To add the interface from scratch in the nomogram dialog, click on the **Insert New Element** button. In this manner you will be creating the interface from the individual elements, the same as creating an interface on the [interface dialog](#). If you already have an interface defined, and wish to clone the elements of that interface for this nomogram, use the button labeled **Clone Elements From** to find the interface and copy the element definition.

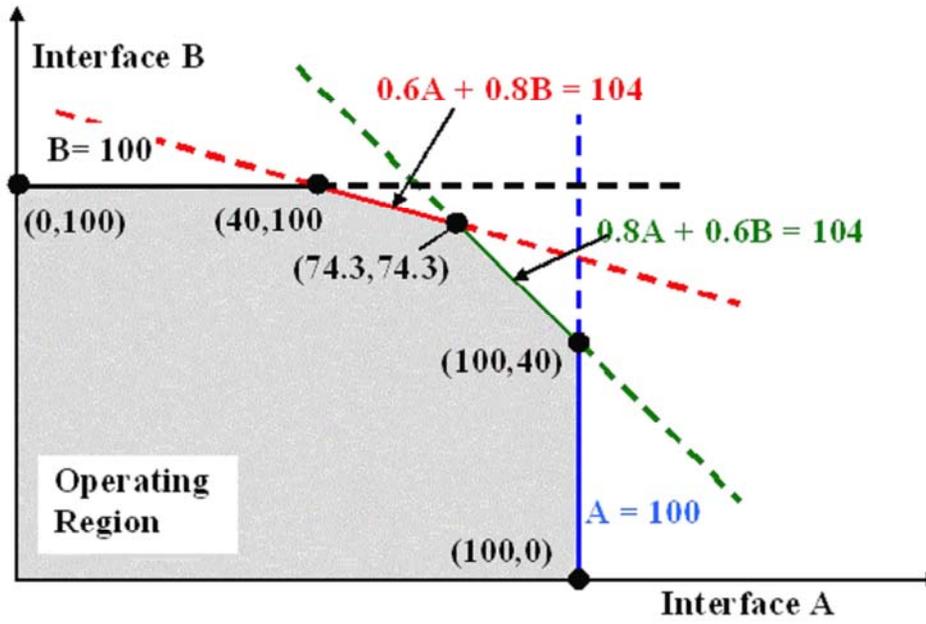
Interface B

The second interface forming the interface pair. To add the interface from scratch in the nomogram dialog, click on the **Insert New Element** button. In this manner you will be creating the interface from the individual elements, the same as creating an interface on the [interface dialog](#). If you already have an interface defined, and wish to clone the elements of that interface for this nomogram, use the button labeled **Clone Elements From** to find the interface and copy the element definition.

Nomogram Breakpoints

This section is used for defining the limit boundaries for the interface. The limit boundaries are defined by inserting nomogram breakpoints. These breakpoints correspond to a pair of MW flows on each interface. In other words, you define the amount of flow allowed on Interface B when interface A is at a certain amount. Typically you will have a flow limit on Interface B that is constant for a certain range of flow in interface A. However, at some point as the flow on Interface A increases, the limit of flow on Interface B can start to decrease due to desired flow limit restrictions of the combined interfaces. At some point, the limit of Interface A would reach a maximum amount and remain constant, and the range of flow on interface B would be fairly small due to the heavy loading in Interface A.

To build this Nomogram Limiting Boundary, begin by right-clicking in the Nomogram Breakpoints list and choose Insert Point. Note that the boundary definition must be a convex piecewise linear curve. You would typically begin by defining the flow limit allowed on Interface B when the flow on interface A is small or zero. Then define breakpoints where the limit on B decreases as the flow on A increases. Eventually you will define a point where the flow on A reaches a limit as the flow on B continues to decrease towards zero. In short, the nomogram limiting boundary is actually a combination of boundary limits that are scaled combinations of the individual interface limits. See the image below for an example of a nomogram limiting boundary.



Injection Group Overview

An injection group is a collection of loads, generators, switched shunts, and/or other injection groups. In that respect, injection groups are somewhat analogous to areas and zones. However, unlike with areas and zones, generators, loads, switched shunts, and injection groups can belong to more than one injection group. Moreover, a single injection group may contain generators, loads, and switched shunts from several different areas and zones. Thus, injection groups are useful when modeling a collection of generators, loads, and switched shunts that act together as a unit, regardless of each individual's area or zone affiliation. The most common use for injection groups is to model a transfer of power from one group of generators and loads to another for [PTDF](#) calculations and for [PV/QV analysis](#). They are called **injection groups** because their components (generators, loads, and switched shunts) are objects that inject power into the network.

Creating Injection Groups

Injection groups can be created from the [Injection Group Display](#), [Injection Group Dialog](#), [Generator Display](#), [Load Display](#), [Switched Shunt Display](#), or by selecting a group of display objects on the online display.

To create an injection group from the Injection Group Display, select **Aggregation > Injection Groups** in the [Model Explorer](#) to open the [Injection Group Display](#). This display is organized into two tab pages with additional tabs and displays available under each. The Injection Groups tab lists and provides a summary of all defined injection groups. Click the right mouse button on this display, and select **Insert** from the resulting [local menu](#) and the [Injection Group Dialog](#) will open with a default name assigned to the new injection group. The Injection Group Dialog allows the addition of participation points to the new injection group and allows updating existing injection groups. The **Load** option on the local menu of the Injection Group Display can also be used to import injection groups from an Injection Group Auxiliary Data File and the **Auto Insert Injection Groups** option can be used to automatically insert injection groups based on options set in the [Auto Insertion of Injection Groups Dialog](#).

If the [Injection Group Dialog](#) is already open, clicking the button labeled **New** will create a new injection group. A prompt will then ask for the name of the injection group to add.

To create an injection group from the [Generator Display](#), [Load Display](#), or [Switched Shunt Display](#), select the elements to add to the injection group from one of these displays, right-click on the selection to bring up the local menu, and select **Create Injection Group from Selection** from the local menu. The [Injection Group Dialog](#) will open with the selected elements added to a new injection group. Any necessary modifications to the group can be made from the Injection Group Dialog.

To create an injection from a selection of display objects, select the objects to add to the injection group. Only generators, loads, and switched shunts will be added to an injection group, but if other objects are selected, the injection group will still be created. The objects that cannot be added to an injection group will simply be ignored. Once all objects are selected, right-click on the selection to bring up the local menu. On the local menu select **Create Injection Group from Selection**. The [Injection Group Dialog](#) will open with the selected objects added to a new injection group. Any necessary modifications to the group can be made from the Injection Group Dialog. More than one object must be selected on the display for the create injection group option to be available.

Auto Insert Injection Groups

Many times injection groups need to be created comprised of generators, loads, or switched shunts grouped by areas, zones, super areas, owners, or some other criteria. Injection groups can be created automatically by setting the grouping options by using the Auto Insertion of Injection Groups Dialog.

The Auto Insertion of Injection Groups Dialog can be accessed by selecting **Records > Auto Insert Injection Groups...** from the case information toolbar on an [Injection Group Display](#). The following options are available:

Element Type

This option is used to select the type of element, **Generators**, **Loads**, or **Switched Shunts**, to add to the new injection groups. Only a single type of element can be added to the new injection groups.

Group By

This option dictates the common property by which all elements in the injection group will be grouped. Elements can be grouped by **Areas**, **Zones**, **Super Areas**, **Owners**, or a selected **Custom Field**. If for example the Area option is selected, then injection groups will be created for each of the defined areas in the case. The same holds true for Zones, Super Areas, and Owners. If the **Custom Field** option is selected, then injection groups will be created for all elements whose Custom Field value is the same. For example, if the Custom Field option is selected and Cust Float 1 is set as the field to use, then all of the elements whose Cust Float 1 field are the same will be grouped together. If there are elements whose Cust Float 1 field is set to 1 and there are elements whose Cust Float 1 field is set to 2, then two new injection groups will be created. One will contain all of the elements whose Cust Float 1 field is 1 and the other will contain all of the elements whose Cust Float 1 field is 2. Elements whose selected Custom Field is blank will not be included in any of the new injection groups.

For **Areas**, **Zones**, **Super Areas**, and **Owners**, the creation of injection groups can be limited by only creating injection groups for which the **Selected** field is set to *YES* for these groups. To use this option, check the **Only Selected** checkbox.

Participation Factor

The options for specifying participation factor are the same as those used on the [Add Participation Points Dialog](#).

Delete Existing Injection Groups

When this option is checked, all existing injection groups will be deleted before inserting the new injection groups.

How to Name the Injection Groups

The new injection groups will be named based on the options set in this section. The **Use Prefix** field determines the prefix that will be used on all injection groups. This field can be blank. The **Start at** field determines what integer to start at when counting the new injection groups. The new injection groups will all be numbered in sequential order starting at the value specified. The **Group By** property chosen will also be part of the injection group name. The **Group By** property can be identified based on **Numbers**, **Names**, or **Both**. The field at the bottom of this section provides an example of how the injection groups will be named based on the option settings.

Do Insert Injection Groups

Click this button to implement the options selected and create the injection groups.

Save to Aux

This saves the option settings specified on the dialog to an auxiliary file.

Deleting Injection Groups

To delete an injection group, select **Aggregations > Injection Groups** from the [Model Explorer](#) to open the [Injection Group Display](#). This display is organized into two tab pages with additional tabs and displays available under each. The Injection Groups tab lists all defined injection groups. Click the right mouse button on this display on the injection group to delete, and select **Delete** from the resulting local menu.

Alternatively, if the [Injection Group Dialog](#) is open, the injection group listed in the Name dropdown box can be deleted by clicking the **Delete** button.

Injection Group Display

The Injection Group Records Display presents data describing each injection group in the case. This display is available by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). This display is organized into two tabs with various sections available under each of the tabs. The tabs and sections provide options for displaying information about injection groups and their associated participation points in a manner in which it is most useful to the user. Injection groups can be displayed with their associated participation points or participation points can be displayed with the injection group to which they belong.

Injection Groups Tab

This tab is organized into a top and bottom section. The top section contains the Injection Groups [Case Information Display](#) that lists all of the defined injection groups. The [local menu](#) is available by right-clicking on the display and contains options available for most case information displays as well as the option to **Show Dialog** to view the [Injection Group Dialog](#).

By default, the Injection Groups Case Information Display contains the following fields:

Name

The name of the injection group. To change the name of an injection group, simply type a new name in the corresponding cell.

Number of Gens

Identifies the number of generators contained in the injection group.

% MW Gen ParFac

Indicates the degree to which generators will contribute to the MW output of the injection group relative to loads. An injection group that has a % MW Gen ParFac value of 100% receives all of its output from generator points; an injection group that has a % MW Gen ParFac of 50% and a % MW Load ParFac of 50% receives equal MW contributions from its constituent loads and generators.

Number of Loads

Identifies the number of loads contained in the injection group.

% MW Load ParFac

Indicates the degree to which MW loads will contribute to the MW output of the injection group relative to generators. An injection group that has a % MW Load ParFac value of 100% receives all of its output from load points; an injection group that has a % MW Load ParFac of 50% and a % MW Gen ParFac of 50% receives equal MW contributions from its constituent loads and generators.

% Mvar Load ParFac

Indicates the degree to which Mvar loads will contribute to the Mvar output of the injection group relative to switched shunts. An injection group that has a % Mvar Load ParFac value of 100% receives all of its Mvar output from load points; an injection group that has a % Mvar Load ParFac of 50% and a % Mvar Shunt ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Number of Shunts

Identifies the number of switched shunts contained in the injection group.

% Mvar Shunt ParFac

Indicates the degree to which switched shunts will contribute to the Mvar output of the injection group relative to Mvar loads. An injection group that has a % Mvar Shunt ParFac value of 100% receives all of its Mvar output from switched shunt points; an injection group that has a % Mvar Shunt ParFac of 50% and a % Mvar Load ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Total MW Injection

Indicates the current total MW injection of all elements in the injection group.

Total Mvar Injection

Indicates the current total Mvar injection of all elements in the injection group.

The bottom section contains additional tabs: **Participation Points (All)**, **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. These tabs contain tables, which are variations of the [Participation Point Records Display](#), that list all of the individual participation points for the selected injection group by element type. Participation points can be added, deleted, or modified for the selected injection group from any of the tables on these tabs. The **Participation Points (All)** table lists only the fields that are relevant for defining participation points while the specific element tables also make available the fields that are specific to that element type.

The **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups** tabs have the option to **Show points that are contained by other injection group points**. This option is only relevant if an injection group contains a participation point that is another injection group. When this option is checked, the individual participation points that are contained in the injection group point will also be displayed. The **Contained by** field will list the name of the injection group that contains a particular participation point. When this option is not checked, the points displayed will be points that are contained in the selected injection group only and no additional information will be provided about the points contained inside any injection group points belonging to the selected injection group. If the **Contained by** field is empty, this means that the participation point is explicitly defined with the selected injection group and does not belong to it through an injection group point.

Participation Points Tab

This tab is organized into a top and bottom section. The top section contains tabs for **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. Tables on these tabs will list power system elements of each particular type. If the option to **Only show elements that belong to at least one injection group** is not checked, then all elements of each type in the case will be listed on one of these tabs. If this option is checked, then only those elements that belong to an injection group will be listed. The tables on these tabs are [Case Information Displays](#) for each particular type and have the full functionality of case information displays.

The bottom section is an Injection Groups Case Information Display. When an element from one of the tables in the top section of the tab is selected, this table will be populated with the list of injection groups to which the selected element belongs.

Injection Group Dialog

The Injection Group Dialog provides information about injection groups and allows their modification. Specifically, the Injection Group Dialog lists the number of generators, loads, and switched shunts contained in the group and the percentage contribution of generators, loads, and switched shunts to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which points can be added and deleted from the injection group's list of participants and the various attributes of the points can be changed.

To view the Injection Group Dialog for a particular injection group, open the [Injection Group Display](#) by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). Find the injection group of interest in the Injection Groups Case Information Display and right-click on it. Then select **Show Dialog** from the resulting local menu.

The Injection Group Dialog contains the following fields and controls:

Name

Identifies the injection group whose information is currently displayed. Selecting a different name from this dropdown box will display information for another injection group.

New, Delete

To insert a new injection group from this dialog, press **New**, and supply the name for the new injection group. To delete the injection group that is currently being shown, click **Delete**.

Save

Changes made to injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click the **Save** button to store any updates with the case.

Gens

Displays the number of generators contained in the injection group.

Loads

Displays the number of loads contained in the injection group.

Shunts

Displays the number of switched shunts contained in the injection group.

% MW Gen Part., % MW Load Part., % MVR Load Part., % MVR Shunt Part.

Displays the relative contributions of generators, loads, and switched shunts to the output of the injection group.

OK, Cancel

Changes made to the injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click **Cancel** to close the dialog without saving the changes. Click **OK** to close the dialog and save the changes.

Participation Points

The tab on the bottom of the Injection Group Dialog lists the points that make up the injection group. This display is called the [Participation Point Records Display](#). By right-clicking on this display, points can be added and deleted from the injection group. Properties of specific points can be changed by

entering changes directly in the table. The **Insert Points** button will open the [Add Participation Points Dialog](#) that can also be used for modifying the points.

Import PTI Subsystems Dialog

Injection groups can be created from PTI subsystem definition files from the [Injection Groups Case Information Display](#). Click the right mouse button on this display and select **Load > Subsystem from *.sub file** from the resulting local menu. From the **Import Injection Groups Dialog**, select the file name for the file to import.

If there is no ambiguity in the import file, new injection groups will be created with the defined subsystem names without further prompts to the user. No ambiguity means that either participation points are defined for buses that have either load or generation but not both or that no participation points are defined and that injection group participation points will be defined based on the maximum generation for each on-line generator in the defined subsystem. If participation points are defined without ambiguity, each participation point will be assigned to either the load or generation at the bus split equally across all loads or generators at the bus.

If there is ambiguity in defined participation points, the Insert PTI Subsystems into Injection Groups Dialog will be displayed. This dialog prompts the user how participation points should be handled for buses with both load and generation or buses with no load or generation.

Buses with Load and Generation

This option allows the user to select how a defined [participation point](#) will be assigned if the participation point bus has both load and generation.

Assign Participation Point to Generation

The participation point will be assigned to generation at the bus. The participation point will be split equally across all generators, either on-line or off-line, at the bus.

Assign Participation Point to Load

The participation point will be assigned to load at the bus. The participation point will be split equally across all loads, either connected or not, at the bus.

Buses with No Load or Generation

This option allows the user to select how a defined participation point will be assigned if the participation point bus has no load or generation.

Ignore Participation Point

The participation point will be ignored.

Add Equivalent Load (closed load with ID='99' and 0 MW and 0 Mvar) and Assign Participation Point to this Load

This option adds a connected load at the participation point bus with ID='99' and 0 MW and 0 Mvar. The participation point is then assigned to this new load.

For each subsystem that is read that is found to have ambiguity in the defined participation points, the Insert PTI Subsystems into Injection Groups Dialog will be displayed unless the user selects Same Options for All Subsystems (Do not prompt again).

Participation Points Overview

A participation point is a member of an injection group. It is a generator, load, switched shunt, or another injection group that participates in, or contributes to, the output of an injection group. Each participation point record identifies the generator, load, switched shunt, or injection group that fills this role, its participation factor, and how that participation factor is calculated. A participation point's participation factor identifies the degree to which the point will contribute to its injection group's output relative to the other points making up the group. Participation factors may be defined as having a fixed value, or they may be re-calculated with every use to stay true to how they were originally defined. For generators, participation factors may be defined as having a fixed value or being equal to the generator's MW reserve in the direction of increasing output (MAX GEN INC), MW reserve in the direction of decreasing output (MAX GEN DEC), or their maximum output (MAX GEN MW).

Each time that an injection group is used, the participation factors for all participation points in the group are normalized so that the factors of all participating participation points sum to 100%. The normalized factors dictate how much a generator, load, or switched shunt contributes to the output of an injection group. The normalization accounts for the re-calculation of the participation factors with each use of the injection groups and the exclusion of participation points because of options set with the various tools that use injection groups. Participation points can be excluded for various reasons including, but not limited to, the exclusion of generator participation points because the generator is not on AGC control or because the generator is at a minimum or maximum limit.

Participation points are added to or deleted from an injection group using the [Injection Group Dialog](#). The Injection Group Dialog houses the [Participation Point Records Display](#), which allows the addition or deletion of points and opening the [Participation Points Dialog](#) to obtain more information about the points.

Participation Point Records Display

The Participation Point Records Display shows information about the points that comprise a particular injection group. This display can be accessed for a particular injection group from the [Injection Group Dialog](#) or the [Injection Group Display](#).

The Participation Point Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which participation points can be deleted or added to the injection group through the [Add Participation Points Dialog](#). Selecting **Insert** from the local menu will open the Add Participation Points Dialog that will allow the addition, deletion, and modification of participation points in the injection group.

The Participation Point Records Display shows the following fields by default:

Point Type

Every participation point is a generator (GEN), load (LOAD), switched shunt (SHUNT), or injection group (INJECTIONGROUP).

Number

Identifies the number of the bus to which a generator, load, or switched shunt is connected.

Name

Identifies the name of the bus to which a generator, load, or switched shunt is connected.

ID

Identifies the ID of the generator, load, or switched shunt or the name of the injection group.

AutoCalc?

If the value of **AutoCalc** is *YES*, the participation factor of the point is re-calculated with every use to be consistent with the way the point's participation factor was initially defined. If the value of **AutoCalc** is *NO*, the participation factor is assumed fixed at its present value.

Initial Value

Indicates how the participation factor of the point was originally computed. For generators, the possible values of this field are

SPECIFIED The participation factor was specified as a constant.

MAX GEN INC The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

MAX GEN DEC The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

MAX GEN MW The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

SPECIFIED The participation factor was specified as a constant.

LOAD MW The participation factor was defined as the size of the load in MW.

For switched shunts, the Initial Value property can assume four possible values:

SPECIFIED The participation factor was specified as a constant.

MAX SHUNT INC The participation factor was defined as the difference between the maximum Mvar output of the switched shunt and its present nominal Mvar output.

MAX SHUNT DEC The participation factor was defined as the difference between the present nominal Mvar output and the minimum Mvar output of the switched shunt.

MAX SHUNT MVAR The participation factor was defined as the maximum Mvar output of the switched shunt.

For injection groups, the Initial Value property can assume only one possible value:

SPECIFIED The participation factor was specified as a constant.

The **Initial Value** field is important if saving the injection groups to an auxiliary file and using them with another case that might have a different generation dispatch or load profile. If the **Initial Value** for a point is specified as *GEN MAX INC*, for example, and the point is loaded from an auxiliary file into another case, Simulator will re-calculate the point's participation factor to match the generator's positive MW reserve in that case.

The **Initial Value** field is also important if the **AutoCalc** field is set to *YES*, because **AutoCalc** uses the rule defined by the **Initial Value** field to recalculate the participation factor with every use. For example, if the **Initial Value** is *MAX GEN INC* and **AutoCalc** is *YES*, the point's participation factor will be updated to match the generator's MW reserve every time the point is accessed.

When the value of the **Initial Value** field is toggled, the point's participation factor, shown in the **ParFac** field, will update to match the new definition. The **Initial Value** field must equal *SPECIFIED* to be allowed to change the value of the point's **ParFac** field by typing the new value directly into the field.

If **AutoCalc** is *NO* and the injection group is not intended to be used with any other case, then the **Initial Value** field should either be ignored or set to *SPECIFIED*.

ParFac

Indicates the participation factor of the participation point. The participation factor defines the relative contribution of the point to the total output of the injection group. The load, generation, or switched shunt change associated with each point is calculated based on the value of the participation factor, with values for points having the largest participation factors experiencing the greatest change.

The participation factor for injection group participation points indicates the relative contribution of the entire injection group to the total output of the injection group to which it belongs. The participation factors of the individual points in the injection group point further define the relative contribution of these points to the total output of the injection group. The following two injection groups provide an example of how injection group points impact the injection group to which they belong when the points are normalized for use:

<u>Injection Group 1</u>	<u>Injection Group 2</u>
Gen 1, ParFac = 50	Gen 5, ParFac = 40
Gen 2, ParFac = 30	Load 1, ParFac = 20
InjectionGroup 2, ParFac = 20	Load 2, ParFac = 20
Total = 100	Total = 80

When Injection Group 1 is used, the participation factors are normalized so that each element provides the following contribution:

Gen 1, Normalized ParFac = $50/100 = 0.5$
 Gen 2, Normalized ParFac = $30/100 = 0.3$
 Gen 5, Normalized ParFac = $20/100 * 40/80 = 0.1$

Load 1, Normalized ParFac = $20/100 * 20/80 = 0.05$

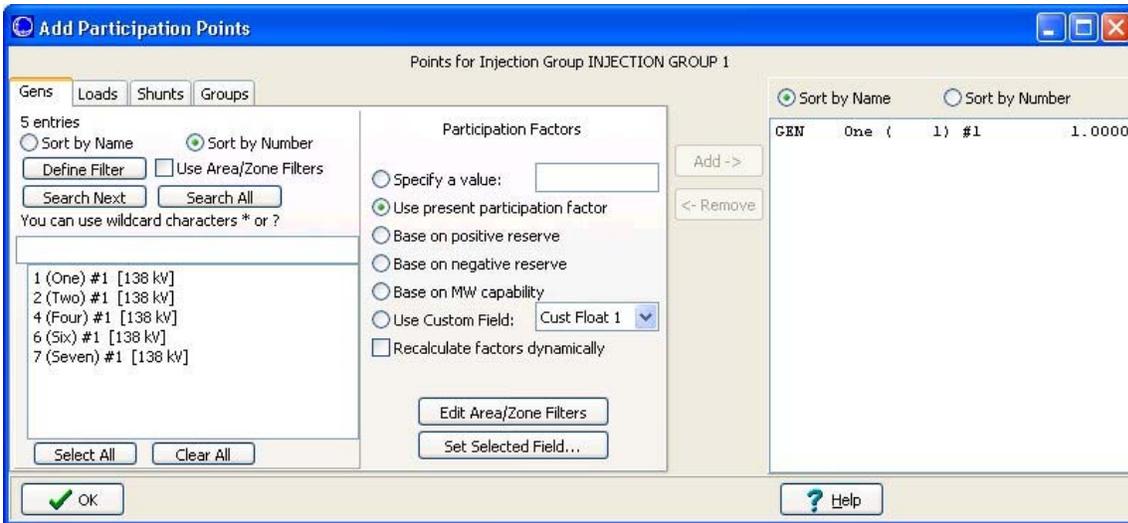
Load 2, Normalized ParFac = $20/100 * 20/80 = 0.05$

Total Normalized = 1.0

To add points to the injection group, right-click on the Participation Point Records Display and select **Insert** from the local menu. This opens the [Add Participation Points Dialog](#).

Add Participation Points Dialog

The Add Participation Points Dialog enables the addition of participation points to an [injection group](#) as well as the modification of existing participation points. This dialog is accessed from a [Participation Point Records Display](#) by right-clicking in the participation points list and selecting **Insert** from the resulting [local menu](#). Participation Point Records Displays are available from the [Injection Group Dialog](#) or [Injection Group Display](#).



The Add Participation Points Dialog features four tabs, one for adding generator points, one for adding load points, another for switched shunts, and a fourth for adding points from injection groups that have already been defined.

The four tabs are almost identical and contain the following controls:

Filtering

If the **Use Area/Zone Filters** box is checked, the list box beneath it, which lists generators, loads, switched shunts or previously defined injection groups depending on the tab, will list only those elements contained in areas or zones or by owners whose [area/zone/owner filter](#) setting is **YES**. If this box is not checked, all generators, loads, and switched shunts in the case will be listed. Injection groups will not be filtered using the area/zone/owner filter.

Alternatively, a custom filter can also be defined by clicking on the **Define Filter** button. This will open the [Advanced Filter](#) dialog, which allows the customization of a filter for determining the devices to list in the display.

Once the list of devices has been set, with or without filtering, the list can be searched using the advanced search techniques in Simulator. These techniques allow the list to be sorted by name or by number and allow the use of wildcard characters. Simply choose Name or Number, and type the name or number of interest in the box. Simulator will look for and highlight the first matching device in the list. If the first device is not the one of interest, use **Search Next** to find the next device that matches the search criteria. For a comprehensive list of all objects matching the search criteria, press **Search All**.

Element List

The box that occupies the left side of each tab lists the generators, loads, switched shunts, or injection groups (depending on which tab is active) that can be added to the injection group. Injection groups will only show up in the element list if they do not cause a circular reference by being added to the current injection group. This means that any injection group that would cause the current injection group to link back to itself will be omitted from the list. Multiple elements can be selected from each of these lists. To select several elements in a row, drag the mouse to highlight the elements to be added. Alternatively, click the first element to add, press and hold the shift key, and click the last element to add. To select elements that are not adjacent in the list, click the first element you to add and hold down the CTRL key while clicking the other elements to add.

Once an element is selected in the Element List, it is ready to be added to the injection group.

Participation Factors

There are several options for defining the participation factors of the selected points.

For generators, a value can be specified, the generator's present participation factor (which comes from the case and is displayed in the [Generator Display](#)) can be used, the participation factor can be calculated as the difference between its present output and either the unit maximum or minimum, its maximum output capability can be used, or the value contained in a selected Custom Field can be used.

For loads, a value can be specified, the participation factor can be based on the load's size, or the value contained in a selected Custom Field can be used.

For switched shunts, a value can be specified, the value contained in a selected Custom Field can be used, or the factor can be based on positive reserve, negative reserve, or MVAR capability.

For injection groups, a value can be specified to use for every point in the injection group, the values already defined for the injection group can be used, or the value contained in a selected Custom Field can be used. These three options are available if the option to **Include Individual Group Points** is checked. When this option is checked, copies of the participation points from the selected injection groups are made and added to the current injection group with the participation factors based on the selected option. When **Include Individual Group Points** is not checked, entire injection groups are included, indicated by the Point Type of *INJECTIONGROUP* in the [Participation Point Records Display](#), in the current injection group and the only option for adding the points is to specify the value for the participation factor. When including an injection group in another injection group as the entire group, the normalization and use of the participation factors follows the discussion of ParFac found in the [Participation Point Records Display](#). Including an injection group in another injection group as the entire group is useful when the injection of the entire group needs to be changed relative to other individual elements or other injection groups.

Recalculate Factors Dynamically

If this box is checked, the participation factors of the points being added will be automatically updated every time the points are used. Such points will then have an **AutoCalc** value of *YES*. If this box is not checked, the participation factors of the points being added will be fixed at the values defined at the time they were created.

To add the points that have been selected, click the **Add ->** button. The new points will be added to the list box on the right.

To update existing points, follow the same process for adding points and simply set the participation factors with the new values. When the **Add->** button is clicked, the existing points will be updated with the new values.

The list box that occupies the right side of each tab lists the points that already comprise the injection group. To delete specific points from the injection group, select them from this list box and click the **<-Remove** button.

To close this dialog, click **OK**.

Chapter 8: Case Information Displays

This chapter describes the case information displays. The case information displays are used to display information about the case, regardless of whether or not it is shown on the onelines.

The following material is included:

- Fundamentals
- Summary Displays
- Area and Zone Displays
- Bus Displays
- Substation Displays
- Generator Displays
- Load Displays
- Line and Transformer Displays
- Switched Shunt Displays
- Interface Displays
- Injection Group Displays
- Island Displays
- Transaction Displays
- Limit Violation Displays
- Ownership Displays
- Load Flow Displays

Model Explorer



Most [Case Information Displays](#) are enclosed inside the Model Explorer. To open the Model Explorer, click on the Model Explorer button shown above. The Model Explorer button is available on both of the [Case Information Ribbon Tab](#) and [Tools Ribbon Tab](#) by default. It is also placed on the [Quick Access Toolbar](#) by default.

Note: normally only one Model Explorer window will be open at a time. If you choose to open a case information display from various places in the Ribbon, the open Model Explorer will be moved to this case information display. If you want multiple Model Explorer to be open, then click on the **Open New Explorer** button below the Explore Pane.

When you open the Model Explorer you will see a case information display. In addition to this the model explorer has 5 additional regions to help you navigate the model as well as modify the model.

Explore Pane

The Explore Pane contains a hierarchical list of most of the objects contained in the power system model. Entries in the Explore Pane that are grayed out represent objects that presently are not defined in the model. The Explore Pane is separated into eight primary folders, with only the Network and Aggregations folder expanded by default. The either primary folders are as follows

- **Recent** - contains a list of case information displays that have been recently opened. This same list of case information displays are depicted on the Recent Tabs across the top of the Model Explorer.
- **Network** - contains a list of the network model objects. Generally these represent actual physical devices such as a generator or transmission line.
- **Aggregations** - contains a list of model objects that represent an aggregation of other model objects. An example is an interface which represents such as interfaces or injection groups
- **Solution Details** - provides access to case information displays that provide details regarding the power flow solution. It includes displays such as the Y-Bus, Jacobian, or summaries of remoted regulated buses
- **Case Information and Auxiliary** - contains generic objects that are operate on the other objects in the model. This includes filtering, expression, Model Criteria and such.
- **Contingency Analysis** - contains a convenient place to access case information displays showing options related to contingency analysis.
- **Fault Analysis** - contains a convenient place to access case information displays showing options related to fault analysis.
- **Optimal Power Flow** - contains a convenient place to access case information displays showing options related to the OPF or SCOPF
- **Transient Stability** - contains a convenient place to access case information displays showing options related to Transient Stability
- **User-Defined** - contains a list of User-Defined Case Information Displays available. These displays are created by right-clicking on the Explore Pane and choosing to Insert User-Defined Case Info. For more information see the [User-Defined Case Information Displays](#) help.

Explore Pane Options

There are a few options regarding the Explore Pane which can be accessed by right-clicking on the Explore Pane, Searchbar, Filterbar, or Case Information Toolbar. When right-clicking a local menu will appear that allows you to specify the following options.

Toolbar (Top)	▶	Fields Pane Location : Allows you to specify the location of the Fields Pane. See the Model Explorer: Fields Pane for more details.
Searchbar (Bottom)	▶	
Filterbar (Top)	▶	
Fields Pane Location	▶	
Save Recent (10)		Save Recent (Number) : Choose this option to change the number of recently viewed case information that are kept and shown as tabs in the Recent Tabs
Show Number of Objects		Show Number of Objects : Check this option to show a number next to each entry in the Explore Pane
Insert User-Defined Case Info		Insert User-Defined Case Info, Remove User-Defined Case Info : These options allow you to insert and remove user-defined Case Information Displays. For more detailed information see the User-Defined Case Information Display help.
Remove User-Defined Case Info		

Fields Pane

The Fields Pane provides direct access to modifying the columns that appear on the case information displays. This replicates the features that can also be accessed by choosing  **Display/Column Options** on the [case information toolbar](#) to open the dialog for [Configuring the Case Information Displays](#). See the [Model Explorer: Fields Pane](#) for more details.

Recent Tabs

The Recent Tabs represent a list of the most recently viewed case information displays on the Model Explorer. You can click on the X to the left of each tab to clear that from the recent list.

Case Information Toolbar

The Case Information Toolbar contain a graphical representation of all the options and actions which can be applied to the case information display. The toolbar is described in detail in the [case information toolbar](#) topic. Also note that all the options displayed on the toolbar are also available by right-clicking on the toolbar and showing the [case information local menu](#).

Case Information Filterbar

The Filterbar provides convenient access to apply an [Advanced Filter](#) or a [Device Filter](#) to the case information display. The toolbar is described in detail in the [Case Information Filterbar](#) topic.

Case Information Searchbar

The Searchbar contains a toolbar that allows you to search the active case information display for particular text. This toolbar behaves in the same manner as the [Search For Text Dialog](#).

A picture of the Model Explorer as it appears when showing the generator records is shown below. Besides showing case information display,

Explore Pane

Case Information Filterbar

Recent Tabs

Case Information Toolbar

Case Information Searchbar

Number of Bus	Name of Bus	ID	Status	Gen MW	Gen Mvar	Set Volt	AGC	AVR	Min MW	Max MW	Min Mvar	Max Mvar	Cost Model	Part. Factor
1	40199 CENTRALA	1	Closed	10.00	0.00	1.02029	NO	NO	0.00	10.00	0.00	0.00	None	10.00
2	40307 COWFALLS	1	Closed	17.00	10.00	1.03913	NO	YES	0.00	35.00	-10.00	10.00	None	35.00
3	40307 COWFALLS	2	Open	0.00	0.00	1.03913	NO	YES	0.00	35.00	-10.00	10.00	None	35.00
4	40769 MV-SVC	1	Closed	0.00	49.00	1.08000	NO	YES	0.00	0.00	49.00	51.00	None	0.00
5	40841 PORT ANG	1	Open	0.00	0.00	1.00725	NO	NO	0.00	12.60	0.00	0.00	None	12.60
6	40841 PORT ANG	2	Open	0.00	0.00	1.00725	NO	NO	0.00	16.00	0.00	0.00	None	16.00
7	42011 ENSERCH1	1	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
8	42012 ENSERCH2	2	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
9	42013 ENSERCH3	3	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
10	42014 ENSERCHL	L	Open	0.00	0.00	1.01700	NO	YES	0.00	58.20	-12.51	30.16	None	58.20
11	42021 SUMAS 1	1	Open	0.00	0.00	1.00000	NO	YES	0.00	83.30	-41.34	61.55	None	83.30
12	42022 SUMAS L	L	Open	0.00	0.00	1.00000	NO	YES	0.00	48.70	-19.03	28.46	None	48.70
13	42031 TENASKA 1	1	Open	0.00	0.00	1.02522	NO	YES	0.00	83.30	-22.36	55.19	None	83.30
14	42032 TENASKA 2	2	Open	0.00	0.00	1.02522	NO	YES	0.00	83.30	-22.36	55.19	None	83.30
15	42033 TENASKAL	L	Open	0.00	0.00	1.02522	NO	YES	0.00	99.90	-38.76	55.32	None	99.90
16	42042 WHITHRN2	2	Closed	74.40	-11.28	0.98478	NO	YES	0.00	74.40	-19.92	47.74	None	74.40
17	42043 WHITHRN3	3	Closed	74.40	-11.41	0.98478	NO	YES	0.00	74.40	-19.92	47.74	None	74.40
18	42111 FREDONA1	1	Open	0.00	0.00	0.98478	NO	YES	0.00	104.00	-66.38	78.12	None	104.00
19	42112 FREDONA2	2	Open	0.00	0.00	0.98478	NO	YES	0.00	104.00	-66.38	78.12	None	104.00
20	42114 FREDONA3	3	Open	0.00	0.00	0.98478	NO	YES	0.00	50.00	-20.00	20.00	None	50.00
21	42115 FREDONA4	4	Open	0.00	0.00	0.98478	NO	YES	0.00	50.00	-20.00	20.00	None	50.00
22	42117 SAWMILL	1	Closed	25.00	7.00	1.01739	NO	YES	0.00	31.00	-7.00	7.00	None	31.00
23	42121 LO BAKER	1	Closed	59.60	1.00	1.02536	NO	NO	0.00	64.00	0.00	2.00	None	64.00
24	42124 UP BAKER	1	Closed	33.91	2.00	1.02681	NO	NO	0.00	47.00	0.00	2.00	None	47.00
25	42124 UP BAKER	2	Closed	33.91	2.00	1.02681	NO	NO	0.00	47.00	0.00	2.00	None	47.00
26	42126 CFALLGEN	5	Open	0.00	0.00	1.00000	NO	NO	0.00	15.00	0.00	0.00	None	15.00
27	42126 CFALLGEN	6	Closed	6.00	0.00	1.00000	NO	NO	0.00	15.00	0.00	0.00	None	15.00
28	42128 KOMO K	1	Closed	7.00	0.00	1.00638	NO	NO	0.00	14.00	-1.00	1.00	None	14.00
29	42131 MRPTGEN1	1	Open	0.00	0.00	1.02500	NO	YES	0.00	38.30	-8.00	20.00	None	38.30
30	42132 MRPTGEN2	2	Open	0.00	0.00	1.02500	NO	YES	0.00	38.30	-8.00	20.00	None	38.30
31	42133 MRPTGEN3	3	Open	0.00	0.00	1.02500	NO	YES	0.00	38.30	-8.00	20.00	None	38.30
32	42134 MRPTGENL	L	Open	0.00	0.00	1.02500	NO	YES	0.00	25.00	-6.00	12.00	None	25.00

Case Information Displays

Simulator offers numerous that provide a convenient, spreadsheet-like view of the power system and its components and are available regardless of whether the case has an associated oneline diagram or not. Many case information displays are available by using the [Model Explorer](#) button found on the [Case Information](#) ribbon tab, although these spread-sheet like views appear throughout the software as stand-alone displays and embedded inside other dialogs. Case information displays are available for buses, bus mismatches, generators, generator costs, ac lines, transformers, transformer impedance correction tables, dc lines, interfaces, areas, zones, schedules, and all other types of objects.

It is also possible to create [User-Defined Case Information Display](#) on the [Model Explorer](#).

Other screens, such as the [Area/Zone/Owner Filters](#) display, also fall into the category of case information displays. There are many different characteristics associated with the Case Information Displays, as discussed in the following topics:

[Colors and Cell Styles](#)

[Using Cell Handles](#)

[Case Information Toolbar](#)

[Case Information Filterbar](#)

[Sorting Records](#)

[Configuring the Case Information Displays](#)

[Finding Records](#)

Configuring the Case Information Displays

The content and format of all case information displays can be controlled through the Display / Column Options dialog. This dialog can be viewed by choosing  **Display/Column Options** on the [case information toolbar](#) or from the case information display's [local menu](#). The features on this display can also be accessed on the [Model Explorer: Fields Pane](#).

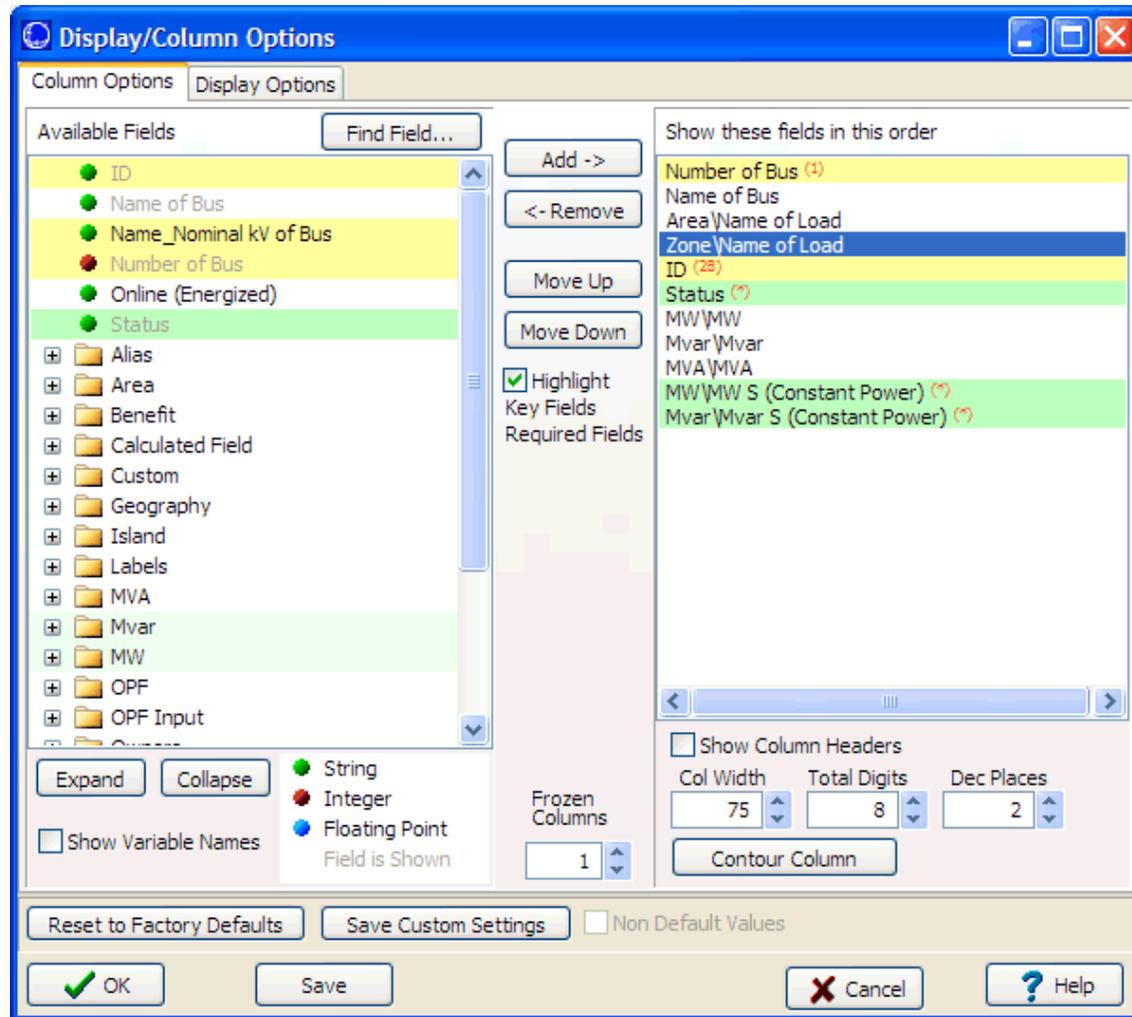
To reset all case information display properties to their default settings, including column and data field associations, click the **Reset All to Default Values** button.

To save your changes and close the dialog box, click **OK**. To save your changes without closing the dialog box, click **Save**. To undo the changes you have specified and close the dialog box, click **Cancel**. Finally, click **Help** to view the corresponding help screen.

While it is normally sufficient to customize the case information displays that comes with the program, it is also possible to create [User-Defined Case Information Displays](#) on the [Model Explore](#). This then provides the ability to switch between different sets of columns, filters, sorting etc... without requiring repeated customizations.

Column Options

The Column Options Tab is shown in the figure below.



Available Fields

This folder view contains all the fields available for addition to the respective case information display. You may select several entries from this list by holding down the Shift and Control keys. Hold down the control key to select several individual fields. Hold down the Shift key to select all fields between two successive mouse clicks. You may then add multiple fields to the Shown fields (see **Add->** below). Fields are generally organized into folder, so make use of the **Find Field** button if you have trouble finding the field you're looking for.

Note: when the mouse hovers over a field, a hint appears giving a longer description of the field.

Find Field

Click this button to search through the list of fields using wild card search.

Expand and Collapse

Click these button to expand or collapse the folder view showing all the available fields.

Show Variable Names

When this box is checked, the variable name corresponding to each column heading will be displayed in the list of available fields alongside the appropriate column heading.

Green, Red, and Blue dots

Also note that fields that the Green, Red, and Blue dots next to each string represents what type of value it is. Also, fields that are shown with gray text in the available field list represent fields that are already shown and in the list on the right.

Show these fields in this order

This list contains the fields which are presently shown on the respective case information display. Not that the field name is representing using backslash \ characters to signify the folders. You may select several entries from this list by holding down the Shift and Control keys. Hold down the control key to select several individual fields. Hold down the Shift key to select all fields between two successive mouse clicks. You may then remove multiple fields (see **Remove** below) or move the order of fields (see **Move** below).

Note: when the mouse hovers over a field, a hint appears giving a longer description of the field.

Column Width

Enter a new value to adjust the physical width of the column. This will affect all fields selected in the Show these fields list.

Total Digits

The number of digits to use when displaying values in the column (including the decimal point.) This will affect all fields selected in the Show these fields list.

Decimal Places

Number of total digits to the right of the decimal point. This will affect all fields selected in the Show these fields list.

Add > (Inserting Columns)

Click the **Add >** button to add all selected available fields to the end of the Show these fields list. You may also click on the selected Available fields and drag and drop the field on the Show these fields list. When drag/dropping the fields, all selected fields will be inserted just before the field that the mouse is over when you drop the fields. Once you have added fields you may then utilize the

< Remove (Removing Columns)

Click the < **Remove** button to remove the fields that are currently selected in the Show these fields list. You may also click on the selected fields in the Show these fields list and drag and drop the fields on the Available fields.

Move Up, Move Down (Moving Columns)

If you selected a block of fields in the Show these fields list, you may change the order of the list by clicking on the **Move Up** or **Move Down** button. You can also move the selection by left-clicking and dragging to move the columns around.

Also note that if you hold the CTRL key down and then left-click and drag a column heading on the case information display, you can move the columns around relative to one another. You will see red highlighting in the column headings to indicate the position to which the field is being moved.

Highlight Key Fields

Checking this option will highlight the *key fields* for the type of object displayed in the case information display presently being modified. The fields highlighted in Yellow in the two lists are either the Primary key fields (numbered) or the Secondary key fields (lettered). Note that some key fields, usually the Circuit ID, are both numbered and lettered, meaning they are used as both Primary or Secondary key fields. Either ALL numbered key fields OR all lettered key fields must be included in the list of shown columns if you intend to export the data to either Excel or a text file, modify it, and paste it back into Simulator.

The fields highlighted in Green are the fields that would be necessary in Excel or an auxiliary file in order to create NEW objects of the type displayed when pasting from Excel or reading in an auxiliary file. If all of the green highlighted fields are not present in the external source, Simulator will not be able to create new objects that may be defined in that source, and will only paste information for objects that already exist in the present case.

Frozen Columns

The number of columns that are fixed and do not scroll when you scroll left or right in the display. Frozen columns will have a background color of gray instead of white. The number of frozen columns is 1 by default.

Display Options

Use Area / Zone Filters

Check this box to restrict the case information display's record set to cover only those areas and zones specified by the [area/zone/owner filters](#). If this box is unchecked, all records, regardless of zone or area, will be displayed. (The area/zone filters option obviously does not apply to the Area/Zone Filters information display.)

Advanced Filter

Clicking this button will open the [Advanced Filters Dialog](#), which will allow more detailed filtering on the display.

Zoom

You can zoom in and out on a case information display by using the keyboard short cut Ctrl+Up and Ctrl+Down. Alternatively you can zoom in out by setting a percentage on this dialog. This can also be modified under the Options Menu of the [Case Information Toolbar](#).

Use Custom Font / Row Height

If checked, then the Row Height and Custom Font specifications will be used. Changing either the Row Height or Custom Font properties will cause this box to become checked automatically. If you wish not to apply your custom specifications to the active case information display, uncheck this box; the default font and row height settings, as defined under the *Case Information Displays* tab of the [Simulator Options](#) dialog box or under the Options Menu of the [Case Information Toolbar](#), will be reapplied.

Row Height

Defines the height of the rows of the case information display.

Change Custom Font

Defines the font with which to display the records.

Set as Default

Sets the current Font and Row Height as the defaults.

Auto Size all Column Widths

Constrains all field widths to contain the widest data elements in each column.

Custom Field Descriptions

Custom Field Descriptions allow specification of custom floating point, integer, and string fields with user-defined column headers and field names. These fields can be added to [case information displays](#) for all power system case data and many supporting data types. Custom fields are currently not available for use with case information displays listing display objects. By default, there are two custom fields of each type available, but any number of each type of field can be defined for each type of data.

To view the Custom Field Descriptions, open **Custom Field Descriptions** found under the **Case Information and Auxiliary** folder on the [Model Explorer](#). The table lists the object types for which custom fields have been defined, how many of each type of field has been defined, and any custom field captions and headers for the fields. By default, there will be three entries; there are two fields each for floating point, integer, and string fields for all data types. These defaults will be used unless specific custom fields are defined for a particular data type.

To insert a new description, select **Records > Insert** from the case information toolbar. To modify an existing description, choose **Show Dialog** from the case information toolbar. Performing either of these actions will open the [Custom Field Description dialog](#). To remove an existing description, choose **Records > Delete**. *Default* descriptions cannot be removed.

The following fields are used for defining the custom field descriptions:

Object Type

Type of object for which the field is available. The *Default* type will be used for any data type that does not have its own definition. This field can only be set using the [Custom Field Description dialog](#) and cannot be changed once a description has been created.

Field Type

The three options for this field are Floating Point, Integer, and String. This is the type of value that will be used in the custom field. This field can only be set on the [Custom Field Description dialog](#) when inserting a new custom field description.

Number of Type

This value determines how many fields of the particular Field Type will be available.

Captions for Field (comma-separated)

Specify the caption to use for each field. These captions will appear in lists of fields for objects such as the list of available fields used with [Display/Column options](#). Enter the captions as a comma-separated list. Leave an empty entry to not assign a particular caption. If a caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Field Captions will be *Integer 1* and *Integer 2*.

If captions are not defined for the fields and Captions for Headers are defined, the header captions will be used instead of the defaults.

Captions for Header (comma-separated)

Specify the caption to use for each field column header. These headers appear in the case information displays for the particular object type. Enter the captions as a comma-separated list. Leave an empty entry to not assign a particular caption. If a header caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Header Captions will be *Cust Int 1* and *Cust Int 2*.

If captions are not defined for the headers and Captions for Field are defined, the field captions will be used instead of the defaults.

Custom Field Description Dialog

The Custom Field Description Dialog is used to create and modify custom field descriptions for a selected object type. This dialog can be accessed from the [Custom Field Descriptions display](#) when inserting a new description by selecting **Records > Insert** or modifying an existing description by selecting **Show Dialog**.

	Field Captions	Header Captions
Floating Point 1	My Caption	My Header
Floating Point 2	Another Caption	Another Header
Floating Point 3	Extra Caption	Extra Header

Object Name

Type of object for which the field is available. Select the type from the drop-down list or use the Find button to search through a list of available types. This field can only be set when creating a new description and cannot be changed once a description has been created.

Field Type

The three options for this field are Floating Point, Integer, and String. This is the type of value that will be used in the custom field. This field can only be set when inserting a new custom field description and cannot be changed once a description has been created.

Number of Type

This value determines how many fields of the particular Field Type will be available.

Clear Captions

Click this button to clear out any previously defined captions.

Field Captions

Specify the caption to use for each field. These captions will appear in lists of fields for objects such as the list of available fields used with [Display/Column options](#). If a caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Field Captions will be *Integer 1* and *Integer 2*.

If captions are not defined for the fields and Header Captions are defined, the header captions will be used instead of the defaults.

Header Captions

Specify the caption to use for each field column header. These headers appear in the case information displays for the particular object type. If a header caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Header Captions will be *Cust Int 1* and *Cust Int 2*.

If captions are not defined for the headers and Captions for Field are defined, the field captions will be used instead of the defaults.

OK, Cancel

Click OK to accept the changes and close the dialog or Cancel to ignore the changes and close the dialog.

Calculated Fields

Calculated fields provide a means of performing an arithmetic operation (minimum, maximum, sum, average, median, variance, standard deviation, count) on a group of objects based on a selected field. Based on the object type of the group, a calculated field can be added to a [case information display](#) for a given object. The calculated field value that is displayed is then based on the calculated field parameters and how the object type in the calculated field relates to the given object.

For example, suppose that you want to find the largest line flow for all lines connected to a bus. A calculated field can be set up using the branch object type with a selected field of MVA (maximum) and the maximum operation can be performed on this field. This calculated field can then be associated with buses by adding this field to the [bus case information display](#). The resulting values for this field will show the maximum MVA flow for all lines connected to a given bus. When determining the value, only the branches that have a terminal bus the same as the given bus will be considered, and the maximum MVA value for all of those branches will be reported.

Optionally, filters can be associated with calculated fields to further specify what objects should be considered in the calculation.

The calculated fields case information display can be accessed from the [Model Explorer](#) under **Case Information and Auxiliary > Calculated Fields**. Existing calculated fields can be modified by selecting **Show Dialog** from the [case information toolbar](#), and new calculated fields can be added by selecting **Records > Insert** from the case information toolbar. Both actions will open the Calculated Field dialog shown below.

Calculated Field Dialog

The screenshot shows the 'Calculated Field' dialog box with the following configuration:

- Calculated Field Name:** Maximum MVA
- Object Type:** Branch
- Branch calculations can be used with the following objects:** 3W Transformer, Area, Bus, Contingency, Contingency Element Block, DC Transmission Line, Generator, Global Contingency Actions, Injection Group
- Field:** MVA\MVA (maximum)
- Operation:** Maximum
- Use Absolute Value:**
- Treat Blank Entries:** As zeros, Ignore
- Objects to be included in the operation:** All, Only objects that meet the filter below
- Select Filter Type:** Branch
- Pre-filter using Area/Zone/Owner Filters:** Enabled (normally checked)
- Logical Comparison:** AND, OR, Not AND, Not OR
- Condition 1:** Find... Choose a Field, between, and
- Buttons:** Add >>, Delete ...
- Bottom Buttons:** OK, Help, Cancel

Calculated Field Name

This is the name of the calculated field as it will appear when added to a case information display. Use the **Save**, **Save As**, **Rename**, and **Delete** buttons to create new calculated fields without exiting the dialog. Switch between existing calculated fields by choosing a name from the drop-down box.

Object Type

Type of object that will be included in the calculation. The type of object determines which objects this field can be used with. This information is displayed to the right of the Object Type option.

Field

Select the field to use in the calculation by selecting a field name from the drop-down box or by clicking the Find button to use [advanced search](#) options to locate the field. Check the **Use Absolute Value** box to use the absolute value of the field in the operation.

Operation

Select the type of operation to perform on the object fields.

Treat Blank Entries

Depending on the field selected, some objects may not have entries. Select to either treat these blank entries **As zeros** or to **Ignore** them in the calculation.

Objects to be included in the operation

This option determines if **All** objects of the selected type will be included in the calculation or if they will be filtered by **Only objects that meet the filter below**.

If choosing to apply a filter, the filter can be defined by using the same options available when defining an [advanced filter](#). The **Set Filter Same As** button can be used to select an existing [advanced filter](#) whose actions should be applied to this filter.

Keep in mind that objects included in the calculation will first be determined by the object type. If for example branch is selected, then all branches will be considered. Setting a filter will limit the branches to only those branches that meet the filter. If the calculated field is used with buses, then the branches considered for a given bus will be only those branches that have a terminal bus the same as the given bus. If the calculated field is used with generators, then the branches considered for a given generator will be only those branches that have a terminal bus the same as the terminal bus of the given generator.

Case Information Customizations Display

To open the Case Information Customizations display, select **Case Information and Auxiliary > Case Info Customizations** from the [Model Explorer](#). If you have customized any of the case information displays for your case, you can see a list of the customized displays and their settings from this display.

Case Information Customization settings cannot be inserted manually or modified on this display. Those modifications must be done on the actual case information display. The primary use of this list display is to provide you a location to save and load case information customizations to an auxiliary file. Customizations can be loaded from an auxiliary file by right-clicking on the grid and choosing to load data from an auxiliary file. To save the customizations, right-click on this display and select **Save As > Auxiliary Data**.

Case Information Displays: Local Menu Options

All case information displays have a set of local menu options. The local menu can be brought up at any time by right clicking on case information display. Most of the actions and options available in the local menu are identical to the options available in the [Case Information Toolbar](#). This is depicted in the figure below.

Number of Bus	Name of Bus	ID	Status	Gen MW	Gen Mvar	Set Volt	AGC	AVR	Min MW	Max MW	Min Mvar	Max Mvar	Cost Model	Part. Factor	
1	ALBURTIS	1	Open	0.00	0.00	1.00000	YES	YES	-9999.00	9999.00	-9999.00	9999.00	Cubic	1.00	
2	30 CONE G1	H	Closed	445.40	41.78	1.06000	YES	YES	262.00	445.40	-21.00	236.00	Cubic	5.45	
3	30 CONE G1	L	Closed	404.60	37.92	1.06000	YES	YES	238.00	404.60	-19.00	214.00	Cubic	4.95	
4	31 CONE G2	H	Closed	445.40	41.78	1.06000	YES	YES	262.00	445.40	-21.00	236.00	Cubic	5.45	
5	31 CONE G2	L	Closed	404.60	37.92	1.06000	YES	YES	238.00	404.60	-19.00	214.00	Cubic	4.95	
6	32 KEYS G1	H	Closed	432.65	168.59	1.05500	YES	YES	254.50	432.65	-21.00	236.00	Cubic	4.81	
7	32 KEYS G1	L	Closed	417.35	152.88	1.05500	YES	YES	245.50	417.40	-19.00	214.00	Cubic	4.64	
8	33 KEYS G2	H	Open	0.00	0.00	1.05500	YES	YES	254.50	432.60	-21.00	236.00	Cubic	4.81	
9	33 KEYS G2	L	Open	0.00	0.00	1.05500	YES	YES	245.50	417.40	-19.00	214.00	Cubic	4.64	
10	34 PCHBTM 2	1	Closed	1110.56	282.69	1.01000	YES	YES	1093.00	1093.00	-319.00	541.70	Cubic	12.80	
11	35 PCHBTM 3	1	Closed	1093.00	282.69	1.01000	YES	YES	93.00	1093.00	-319.00	541.70	Cubic	12.80	
12	36 SALEM G1	1	Open	0.00	0.00	1.06000	YES	YES	0.00	1106.00	-250.00	550.00	Cubic	13.00	
13	37 SALEM G2	1	Closed	1119.00	282.69	1.01000	YES	YES	0.00	1106.00	-250.00	550.00	Cubic	13.00	
14	38 SUSQ 2	1	Closed	1144.00	282.69	1.01000	YES	YES	44.00	1144.00	-258.00	560.00	Cubic	12.80	
15	39 HOPE CG1	1	Closed	1043.00	282.69	1.01000	YES	YES	81.00	1071.00	-250.00	550.00	Cubic	13.00	
16	40 C CLF G1	1	Closed	838.00	282.69	1.01000	YES	YES	85.00	838.00	-150.00	400.00	Cubic	10.20	
17	41 C CLF G2	1	Open	0.00	0.00	1.06000	YES	YES	80.00	838.00	-65.00	400.00	Cubic	10.03	
18	42 LIMERCK2	1	Closed	1150.00	282.69	1.01000	YES	YES	50.00	1150.00	-442.00	276.00	Cubic	12.65	
19	204 C.SLOPE	1	Closed	88.00	282.69	1.01000	YES	YES	88.00	88.00	0.00	50.00	Cubic	1.02	
20	204 C.SLOPE	2	Closed	50.00	282.69	1.01000	YES	YES	50.00	50.00	0.00	90.00	Cubic	0.60	
21	295 WRREN #1	1	Closed	41.00	282.69	1.01000	YES	YES	16.00	41.00	-1.00	9.50	26.00	Cubic	0.63
22	307 WRREN #2	1	Closed	41.00	282.69	1.01000	YES	YES	16.00	41.00	-1.00	9.50	26.00	Cubic	0.48
23	312 PINEY #1	1	Closed	9.00	282.69	1.01000	YES	YES	9.00	9.00	-13.50	18.00	Cubic	0.12	
24	312 PINEY #1	2	Closed	9.00	282.69	1.01000	YES	YES	9.00	9.00	0.00	9.00	Cubic	0.12	
25	312 PINEY #1	3	Closed	9.00	282.69	1.01000	YES	YES	9.00	9.00	0.00	9.00	Cubic	0.12	
26	318 GNERAL E	1	Closed	24.00	282.69	1.01000	YES	YES	26.00	24.00	4.00	4.00	Cubic	1.00	
27	319 HAMMERML	1	Closed	31.00	282.69	1.01000	YES	YES	38.00	31.00	6.50	6.50	Cubic	1.00	
28	334 TITSVL #1	1	Closed	25.00	282.69	1.01000	YES	YES	0.00	25.00	-6.00	10.00	Cubic	1.00	
29	335 WAYNE CT	1	Open	0.00	282.69	1.06000	YES	YES	56.00	56.00	-25.49	39.49	Cubic	0.63	
30	336 WARRN CT	1	Open	0.00	282.69	1.06000	YES	YES	57.00	57.00	-3.00	23.00	Cubic	0.68	
31	343 ILP CO-G	1	Closed	16.00	282.69	1.01000	YES	YES	16.00	16.00	4.00	4.00	Cubic	0.30	
32	349 SENECA #1	1	Closed	210.00	282.69	1.01000	YES	YES	10.00	210.00	-78.29	116.32	Cubic	2.20	
33	350 SENECA #2	1	Closed	195.00	282.69	1.01000	YES	YES	17.00	197.00	-69.00	97.18	Cubic	2.20	

For help on most of the actions and options available in the local menu, please see the [Case Information Toolbar](#) help. The other options that are also sometimes available on the right-click menu however are listed below.

Why is this field or line disabled?

This option will display a message showing the reason why the field is disabled. This option will be enabled only if the field is actually disabled.

Help

Display context-sensitive help for the case-information display.

Form Control

This menu option allows to control the case information display.

Print

You can print the contents of most of the case information displays by selecting Print from the local menu.

Close

Closes the case information display.

Make Top Left of Form Visible

Makes the top or the left of the case information display visible, if either the top or the left are not visible.

Shift Form Up

Moves the case information display up so the bottom of the case information display can be visible.

Maximize

Maximizes the size of the case information display.

Unmaximize (Restore)

Restores the size of the case information display.

Minimize

Minimizes the size of the case information display.

Case Information Displays: Colors and Cell Styles

The case information displays distinguish data field types by text color and cell style. Most of the entries on the case information display are colored using the following color convention. These colors can be customized using the Case Information Displays tab on the [Simulation Options Display](#), which can be invoked by selecting **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. **Simulator Options** is also available on the [Tools](#) and [Case Information](#) ribbon tabs.

There are three types of data fields:

Standard Fields Fields that cannot be modified directly from the case information display are colored black by default.

Enterable Fields Fields that can be modified are colored navy blue by default. When selected, a cell containing an enterable field will display a tiny filled square in its bottom right corner. This box is called a cell.

Toggleable Fields Fields whose values can be toggled are colored green by default. The values contained in toggleable fields are modified by left-clicking on them. Like cells in enterable fields, cells in toggleable fields display a cell handle when selected.

Besides indicating field type, color coding is also used to highlight violations of branch flow, generator MW or MVR output, and bus voltage constraints. Fields that are either at a limit or violating a limit are colored red by default. This, too, is configurable from the Case Information Displays tab on [Simulation Options Display](#).

Case Information Displays: Using Cell Handles

When selected, cells corresponding to enterable and toggleable fields exhibit a small filled square in their bottom right corner called a cell handle. The cell handle may be used to propagate the value of the selected cell to other cells in the same field.

Suppose we have selected a toggleable or enterable cell and that we wish to copy its value to other records. Call this cell the *source cell* and its value the *source value*. To copy the source value to another record or records, perform the following steps:

- Drag the mouse onto the cell handle until the pointer becomes a crosshair.
- With the mouse pointer showing as a crosshair, click and hold the left mouse button.
- With the left mouse button depressed, drag the mouse up or down from the source cell to select a group of records to which to copy the source value. These destination cells will display a yellow background when selected in this manner.
- When you have finished selecting the destination cells, release the left mouse button.
- A message box will appear asking whether you want to change the values of the destination cells to the source value. Answer Yes to complete the copy.

Case Information Displays: Sorting Records

You can sort the entries on the case information displays by just about any field. To sort the records by a particular column, left-click on the column's heading. Left-click the column's heading again to reverse the sort order.

To sort the records by the absolute value of a field, hold down the shift key as you left-click on the field's heading.

Simulator also has a more advanced sorting tool, which can be accessed by selecting **Advanced Sort** from the [local menu](#). Advanced sort allows you to sort information based on values in more than one column of data. Advanced sort also allows you to sort based on the absolute value of numerical fields and by case sensitivity for string fields.

Case Information Displays: Finding Records

The Find Dialog is used to find the device of the specified type on the different Case Information displays. The Find Dialog is available on most case information displays from the local menu, which can be invoked by right-clicking in the grid.

Different Find dialogs exist for Areas, Buses, Interfaces, Lines and Zones. Note that the Bus Dialog is used to locate all bus objects, such as generators, loads, switched shunts and the bus itself. In general, the format of each dialog is similar, allowing you to find the desired object using either its number or name. The only exception is interfaces, for which no name is defined. The basics of the Find dialog are explained in the [Find Dialog Basics](#) help topic.

Saving Case Information Display Contents as HTML Tables

Simulator's [Case Information Displays](#) allow you to save their contents as HTML tables for display on the world-wide web. To do this, right-click on any row of the table and select **Save As HTML** from the resulting local menu. This brings up the [Table Format Dialog](#). Set the various table formatting options and click **OK**. Then, select the name of the file to which to save the HTML code. Finally, if a region of the table was selected, you will be asked if you want to save just the selected region as HTML. Indicate **Yes** to convert just the selected portion of the table, or click **No** to write the entire table as HTML.

HTML Table Format Dialog

The Table Format Dialog allows you to set various formatting options for HTML tables. This form is invoked when you try to save a [Case Information Displays](#) as HTML.

The Table Format Dialog is divided into two tabs, **Table Properties** and **Table Elements**.

Table Properties

This tab allows you to specify values for options that govern the appearance of the table as a whole. Here you can specify the following properties:

Border Weight

The thickness of the border to draw around each cell. Specify 0 to suppress the drawing of a cell border.

Horizontal Cell Spacing

The spacing to employ between cells that neighbor each other horizontally.

Vertical Cell Spacing

The spacing to employ between cells that neighbor each other vertically.

Table Width

The width of the table. If the Percent checkbox is checked, the width specifies the horizontal dimension of the table relative to the object that contains it on the screen.

Caption

The table title that will be printed directly above it on the web page.

Table Elements

These options control how the data will be centered in each cell. Choices include:

Horizontal Alignment

Controls how the text in each cell should be positioned horizontally. The default value is Left, but you may also choose to center or right-justify the data in each cell.

Vertical Alignment

Controls how the text in each cell should be positioned vertically. The default value is Middle, but you may also choose to align the text with the top or bottom edges of the cell.

If any of the numeric entries (Border Weight, Horizontal Cell Spacing, Vertical Cell Spacing, Table Width) are left blank or zero, your browser will employ its default settings for these values in rendering the table on the screen.

Set contains actions for setting data or toggling data.

Columns contains actions that act on the column such as [Get Column Metrics](#). For more details see the [Case Information Toolbar: Set, Toggle, and Column Menu Help](#).

Copy, Paste and Send Menu

The Copy, Paste and Send Menu contains actions that copy and paste data through the Windows Clipboard. In addition it contains actions that send data directly to spreadsheet software such as Microsoft Excel or Open Office Calc. For more details see the [Case Information Toolbar: Copy, Paste, and Send Menu Help](#).

Save Auxiliary Menu

The Save Auxiliary Drop-Down contains actions that save data in the case information display to various formats. The [Auxiliary File Format](#) and Comma-Separate-Variable (CSV) formats are always available, but some records show other more specific formats. For more details see the [Case Information Toolbar Save Auxiliary Files Menu Help](#).

Load Auxiliary Menu

The Load Auxiliary Drop-Down contains actions that load data records related to the active case information display. The [Auxiliary File Format](#) is always a choice, but some records show other more record-specific formats. For more details see the [Case Information Toolbar: Load Auxiliary Files Menu Help](#).

Area/Zone/Owner Filters

Opens the [Area/Zone/Owner Filters](#) display which allows filtering of the case information displays by Area, Zone, and Owner.

Filtering Menu

The Filtering Drop contains actions related to filtering the records shown in the active case information display. For more details see the [Case Information Toolbar: Filtering Menu Help](#).

Advanced Sort

Allows the user to custom sort the information in the display based on desired criteria. See [Case Information Display: Sorting Records](#) for more information.

Refresh Display

Select this option to update the currently displayed data to match the present state of the system.

Options Menu

Invokes a drop down menu similar to the Case Information Display options dialog. See [Case Information Display Options](#) for more details.

Highlight Key and Required Field Column Headings

If this option is checked, the key fields needed for identifying objects will be highlighted in the case information display. These fields are important if you are planning on pasting data back into Simulator from Excel or reading data in from an auxiliary file. If the necessary key fields are not present in the Excel paste of auxiliary file, the data in those formats cannot be processed by Simulator.

Similarly, the Required Field Column Headings can also be highlighted in the case information display. In order for Simulator to create new objects by reading them from an auxiliary file or and Excel paste, the required fields must be present in the pasted data or the object cannot be created.

Headings

This option allows you to choose whether the column headings of the case information displays are the normal column headings or are the variable names of the type of data stored in each column. Variable names are the strings used when writing data out to an [Auxiliary File](#). Change the headings to variable names can be helpful when looking at or creating auxiliary files.

Show Grid Lines

Grid lines are shown by default in case information displays. Unchecking this box will remove the grid lines from the displays.

Show Header Hints

This option allows the hints for the current field to be displayed when moving the cursor over the column heading.

Word Wrap Headings

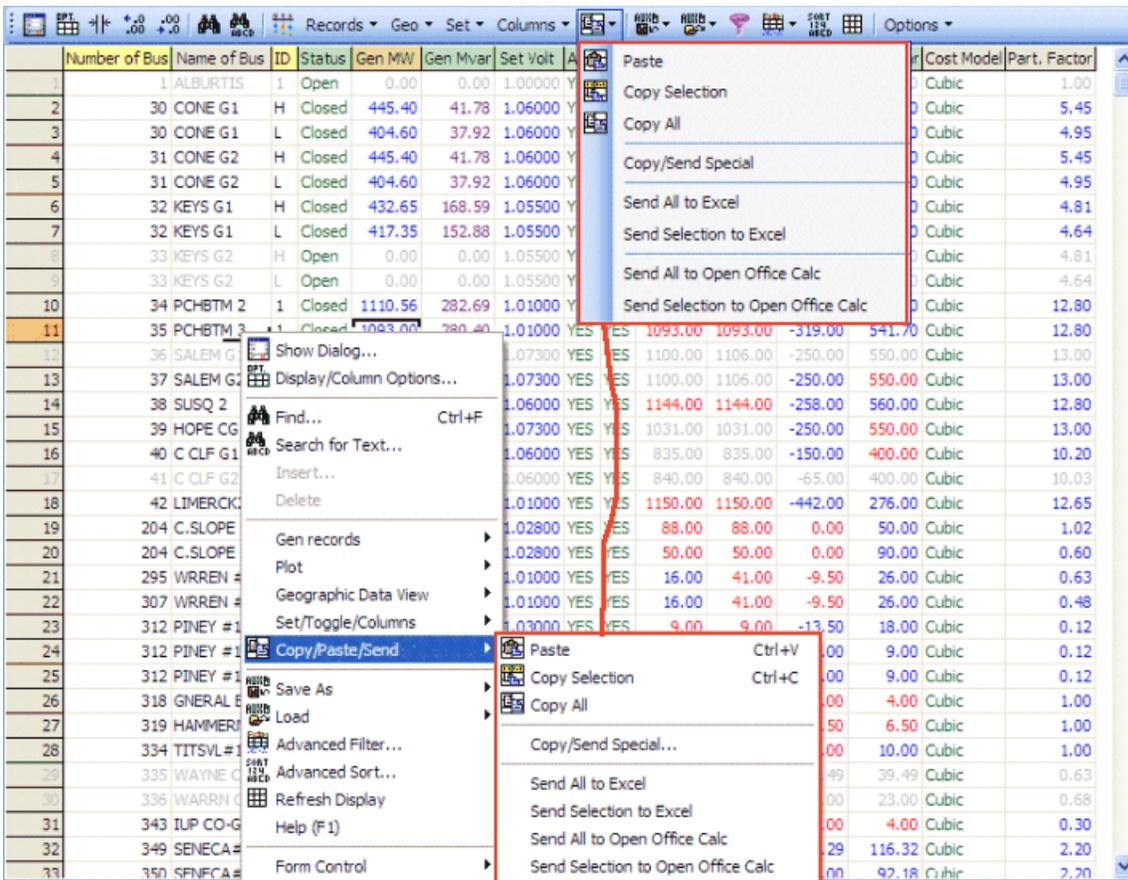
Check this box to wrap the heading column text, breaking it in several rows to fit the current column width. If unchecked, the column heading text will be displayed in only one row.

Highlight Objects if Selected? Field = YES

Check this option to automatically highlight all case information rows that represent a model object that has its "Selected" field set to YES. The row will then be highlighted using the color specified on this dialog. This option will apply to all case information displays representing all types of objects.

Case Information Toolbar: Copy, Paste and Send Menu

The [Case Information Toolbar](#) Copy, Paste and Send Menu displays the identical options as shown in the Copy/Paste/Send menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



Paste

Select Paste from the local menu to copy a record set from the Windows clipboard into the case information display. See [Copying Simulator Data to and from Other Applications](#) for more details.

Copy Selection

The Copy Selection menu option copies the records selected in the case information display to the Windows clipboard, from which the selection can be copied into other programs such as Microsoft Excel for further analysis. See [Copying Simulator Data to and from Other Applications](#) for more details.

Copy All

The Copy All menu option copies the entire record set contained in the case information display to the Windows clipboard, from which it can be copied into other programs such as Excel for further analysis. See [Copying Simulator Data to and from Other Applications](#) for more details.

Copy / Send Special

The Copy / Send Special menu option will open the a dialog that allows the user to set a few custom options before completing the data copy. The user can choose to copy all or a selection, and to copy the data either to the Windows clipboard or send it directly to Excel. Also, you can choose to copy or

send the transpose. In addition, the user can choose whether to use the normal column headings or variable names (see the list of fields in [Data Argument List for Auxiliary Files](#)) as column identifiers. Lastly, the user can specify whether or not to include a row containing the type of object the data represents (Object Name) and a row containing the column headers for each column of data. Note that for pasting the information back into Simulator, the Object Name and Column Heading rows must be contained with the data to be pasted. If you have changed the settings on this dialog and wish to make them the case default settings, click on the Make Default for all Copy Actions button.

Send All to Excel (or to Open Office Calc)

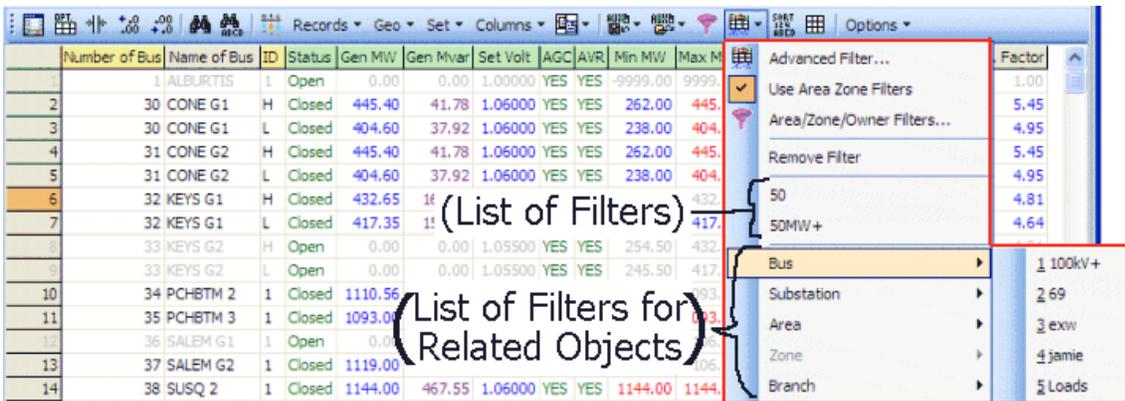
The Send All to Excel menu option copies the entire record set contained in the case information display and automatically sends it to Excel. The first time this option is selected, Simulator will start a new instance of Excel on your machine and paste the data on the first sheet. Subsequent calls to Send to Excel will continue to add sheets and paste data to this instance of Excel, until the Excel instance is closed manually by the user. This functionality works identically for the Open Office Calc spreadsheet as well.

Send Selection to Excel (or to Open Office Calc)

The Send Selection to Excel menu option copies the selected record set in the case information display and automatically sends it to Excel. The first time this option is selected, Simulator will start a new instance of Excel on your machine and paste the data on the first sheet. Subsequent calls to Send to Excel will continue to add sheets and paste data to this instance of Excel, until the Excel instance is closed manually by the user. This functionality works identically for the Open Office Calc spreadsheet as well.

Case Information Toolbar: Filtering Menu

The Filtering Menu, contained on the [Case Information Toolbar](#), gives access to several options regarding any [Advanced Filters](#) that can be applied to the active case information display. In addition it provides access to the [Area/Zone/Owner Filters](#). The drop-down menu is shown in the following figure.



The top of the menu has four primary buttons that appear on all case information displays. These four are as follows

Advanced Filter

Allows the user to custom filter the information in the display based on desired criteria. See [Advanced Filtering](#) for more information.

Use Area/Zone/Owner Filters

Filters displayed records by selections made on the [Area/Zone/Owner Filters](#) display.

Area/Zone/Owner Filters

Opens the [Area/Zone/Owner Filters](#) display for setting the areas, zones or owners for which the filter should apply.

Remove Filter

Click this button to remove the Advanced Filter from the present case information display. This does not delete the Advanced Filter definition, but only removes the reference to it from the active case information display. The filter can easily be reapplied after removing it. If you would like to permanently delete an Advanced Filter see [Advanced Filtering](#).

After the top section of the menu are two sections that may or may not be available for a particular case information display

(List of Filters)

This contains a list of advanced filters for the object type displayed in the active case information display. These filters would have already been created using either the [Advanced Filtering](#) dialog or by reading them from an [Auxiliary File](#). In the example image above the "50" and "50MW+" entries represent filters that apply to a generator object. If there are more than 10 advanced filters for the present object type, then a submenu will be shown for the object type as well. Also if an advanced filter is presently applied to the case information display, then the active advanced filter will be denoted with a check mark.

(List of Filters for Related Objects)

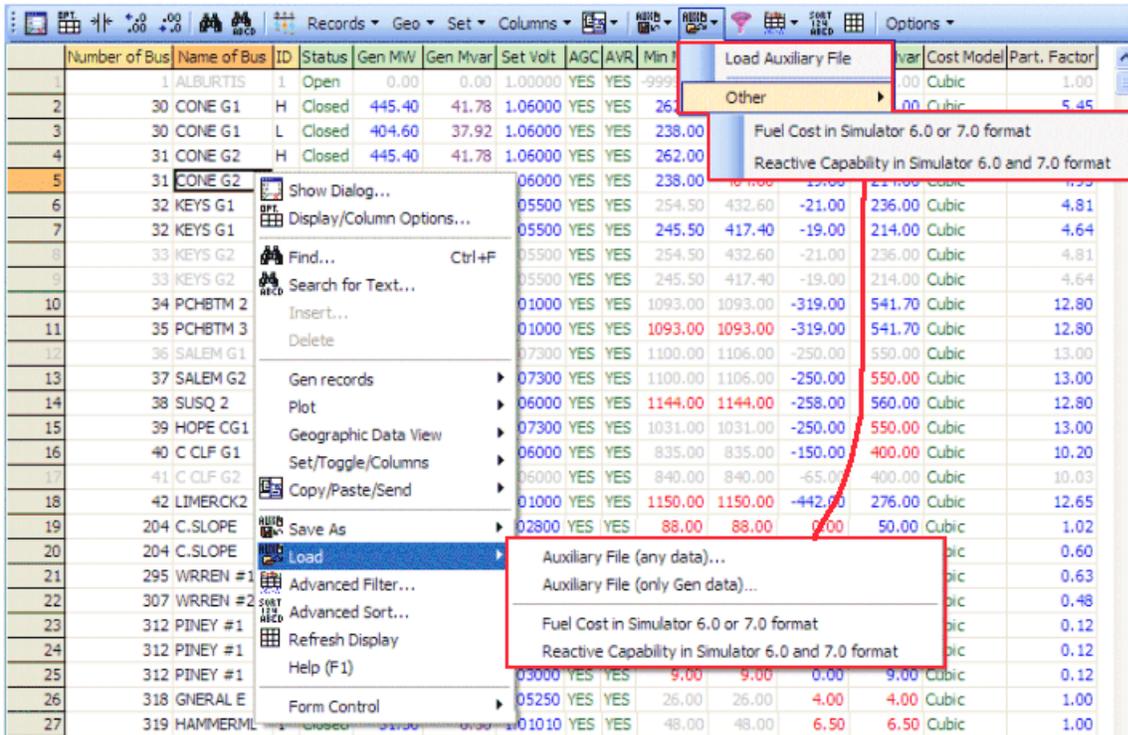
This contains a list of advanced filters whose object type is *related* to the type of record displayed in the active case information display. For example, the case information display shown in the figure above is showing generator records. Generator records allow you to filter the generator object using a Bus, Substation, Area, Zone, or Branch Filter. In the example above there are 5 filters defined for Bus objects (100kV+, 69, exw, jamie, and Loads) which can be used to filter the generator records. For more information on Related Object Filtering, see the [Advanced Filtering](#) dialog. Also if an advanced filter is presently applied to the case information display, then the active advanced filtered will be denoted with a check mark and the object type of that filter will be denoted with a grayed out check mark.

Case Information Toolbar: Geo Data View Menu

The Geo Menu, found on the [Case Information Toolbar](#), contains options related to opening the Geographic Data View Displays based on the objects selected in the case information display. For more information see the help on [Geographic Data Views](#).

Case Information Toolbar: Load Auxiliary Files Menu

The [Case Information Toolbar](#) Load Auxiliary Menu displays the identical options as shown in the Load menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



This option will load the contents of an external file to the case information display.

Auxiliary File (Any Data)

This option allows loading an auxiliary file containing any data.

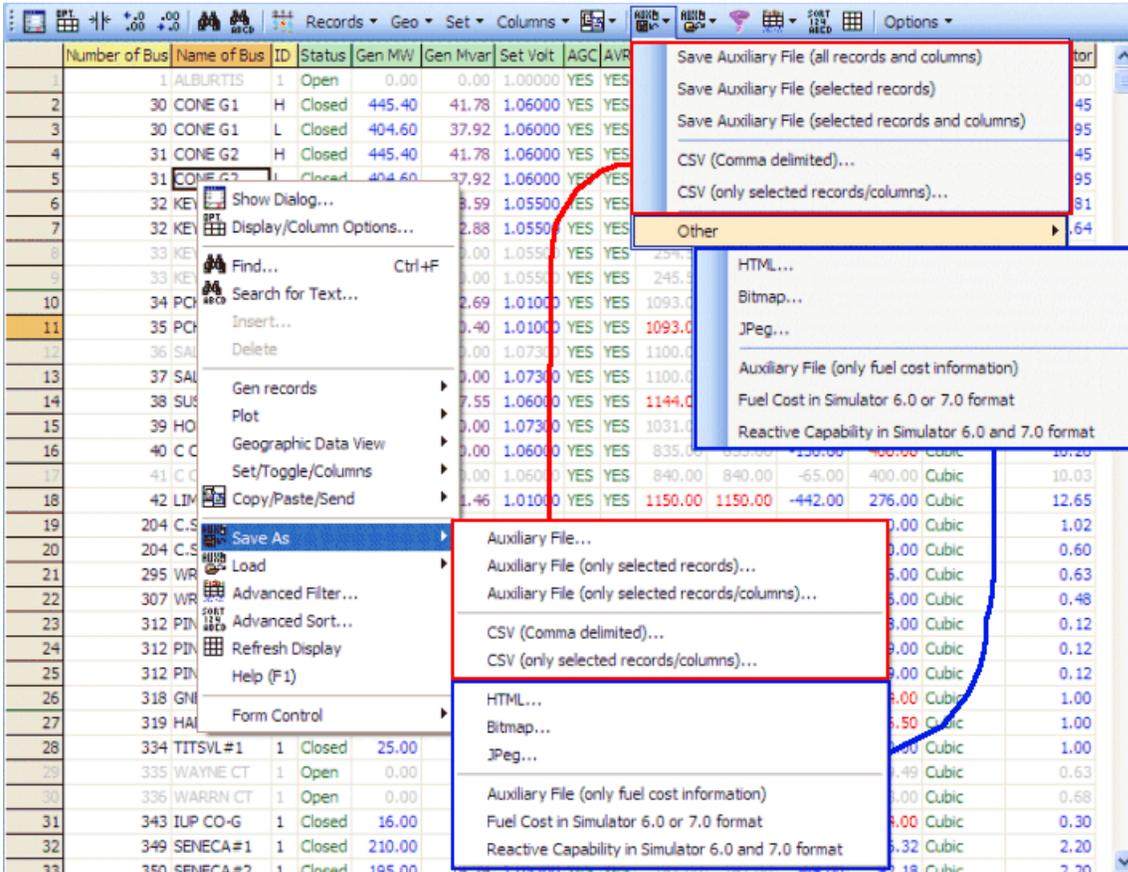
Auxiliary File (Only *Specific* Data)

This option allows loading an auxiliary file containing only data related to the current case information display.

At the end of the menu, some case information displays show more specific formats that are relevant only for that type of record. For instance in the figure above you can see support for loading in older file formats previously supported.

Case Information Toolbar: Save Auxiliary Files Menu

The [Case Information Toolbar](#) Save Auxiliary Menu displays the identical options as shown in the Save As menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



This option will save the contents of the case information display to an external file.

Auxiliary File

This option allows saving all the contents of the case information display to an auxiliary file.

Auxiliary File (only selected records)

This option allows saving the selected record set in the case information display to an auxiliary file.

Auxiliary File (only selected records/columns)

This option allows saving the selected record set and the selected columns in the case information display to an auxiliary file.

CSV (Comma Delimited)

This option allows saving all the contents of the case information display to a comma delimited file.

CSV (only selected records/columns)

This option allows saving the selected record set and the selected columns in the case information display to a comma delimited file.

HTML

You can save the entire table or selected records to an HTML file for viewing from an Internet browser. For more information, see [Saving Case Information Display Contents as HTML Tables](#).

Bitmap

This option allows saving the case information display into a Bitmap picture file.

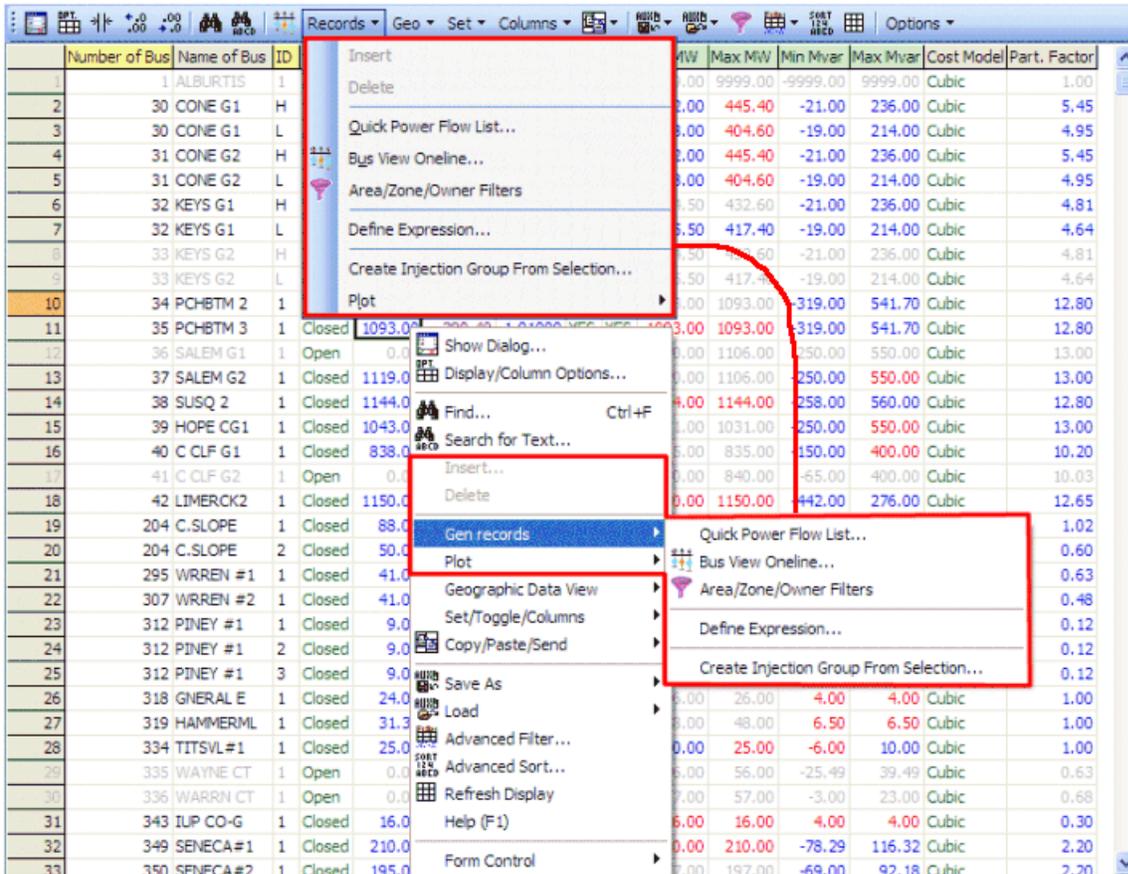
Jpeg

This option allows saving the case information display into a JPEG picture file.

At the end of the menu, some case information displays show more specific formats that are relevant only for that type of record. For instance in the figure above you can see a specific Auxiliary File format options which automatically saves all the data related to the cost information of the generator.

Case Information Toolbar: Records Menu

The [Case Information Toolbar](#) Records Menu displays a combination of the Insert/Delete buttons and the Records and Plot menus on the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



Insert and Delete

Many records have an Insert and Delete related buttons. These are used for inserting new records and deleting records. Note that some records can only be inserted or deleted in Edit Mode. In such cases the Insert and Delete button will be grayed out when not in Edit Mode. Use the Bus View Online local menu option

Records Menu

Quick Power Flow List

Use the Quick Power Flow List local menu option to invoke [Simulator's Quick Power Flow List](#) tool.

Bus View Online

Use the Bus View Online local menu option to bring up the [Bus View Display](#), which illustrates how the selected bus is connected to the rest of the system.

Substation View Online

Use the Bus View Online local menu option to bring up the [Substation View Display](#), which illustrates how the selected substation is connected to the rest of the system.

Define Expression

Allows you to define [Custom Expressions](#) that are functions of other fields.

Pan to Object on Open Onelines

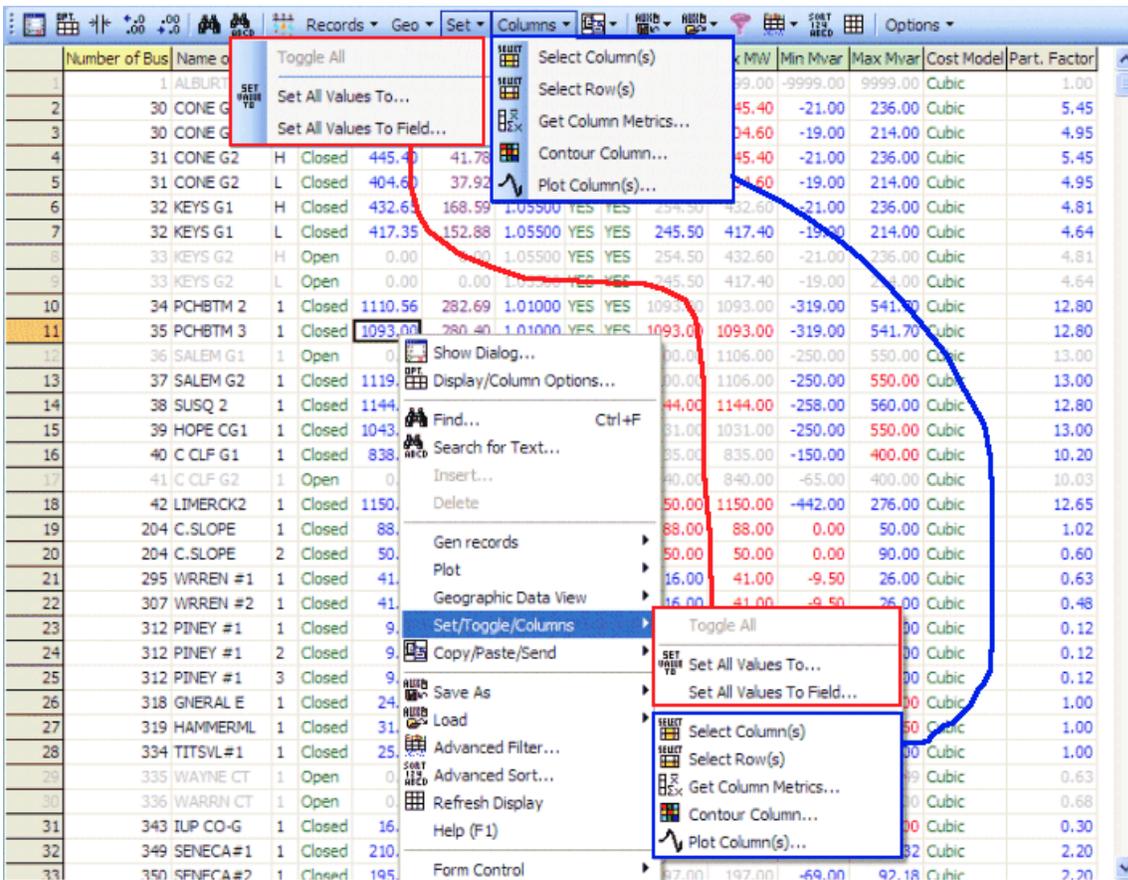
Selecting this option will move the view on any open oneline diagrams to focus on the device clicked in the table, if an object representing the device exists on the diagram.

Record-Specific Actions

At the end of the Record Drop-Down will be a list of options that are special for the particular kind of record shown in the active case information display. For example, in the image above for a generator case information display the option "Create Injection Group From Selection..." allows you to create a new injection group containing all of the presently selected generator records.

Case Information Toolbar: Set, Toggle, and Columns Menus

The [Case Information Toolbar](#) Set and Columns Menus display the identical options as shown in the Set/Toggle/Columns menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



Set All Values To

Click on this menu item to bring up a dialog which allows you to set all the values in a column to the same value. For instance you might change all the buses to have an angle of zero degrees.

A special trick that can be used in conjunction with this option is entering a string of the form "@variablename". When entering data into a field, if the string begins with the symbol @, then Simulator will parse the remaining part of the string to see if it matches one of the variable names associated with that type of object. Variable name strings are what are used in the Auxiliary File Format (see the list of fields in [Data Argument List for Auxiliary Files](#)). Essentially this trick allows you to copy one column to another column. Using this trick, you could change the set-point voltage of all the generators in the case to be equal to the terminal bus voltage of the generator.

Set All Values To Field

Click on this menu item to bring up a dialog which allows you to select a field of this record type and then set all the values in a column to the this field. For instance you might change all the generator setpoint voltage to be equal to their present regulated bus voltage. This represents a user-interface method of employing the special trick using the "@variablename" string as discussed in the Set All Value To comments above.

Select Column(s)

Use this option to quickly select entire columns in the grid. First highlight the cells of a single record for the columns you wish to highlight, and then choose this option to highlight the entire column.

Select Row(s)

Use this option to quickly select entire columns in the grid. First highlight the cells of a single record for the columns you wish to highlight, and then choose this option to highlight the entire column.

Get Column Metrics

This option allows you to compute the metrics for the selected column. Choosing Get Column Metrics from the local menu will bring the [Grid Metrics Dialog](#). This option is only available for columns whose content is numeric.

Contour Column

Use the Contour Column local menu option to contour a column of the list display. Choosing Contour Column from the local menu will open the [Contour Column Dialog](#).

Plot Column(s)

Select this option to plot the values in any selected columns.

Area/Zone/Owner Filters

The Area/Zone/Owner Filters Dialog allows you to filter the information shown on the [case information displays](#) and other dialogs by area, zone or owner. There are a several ways to open the Area/Zone/Owner Filters Dialog.

- Go to the [Case Information](#) ribbon tab, and choose  **Area/Zone Filters** from the **Case Information** ribbon group
- Click on the button  on the [Case Information Toolbar](#)
- There are also many buttons on user interface dialogs throughout the program that invoke this dialog as well.

For small cases, you will usually not need to use this filtering capability, but it can be very useful for large cases. The filters display lists each area, zone and owner in the case, the number of buses in each, the range of bus numbers contained in each, and whether or not information about that area, zone or owner should be displayed. In order for a device to be displayed, its area, zone and owner Shown property must be set to yes. For devices which can belong to more than one area, zone, or owner (for instance a transmission line that connects two areas, i.e. a tie line), then the device will be displayed if any end of the device meets the area/zone/owner filter.

You can switch between displaying the filters for the case areas or the zones by clicking on the associated tab.

The area/zone/owners filters list is itself a case information display and therefore shares many of the same [local menu](#) options and characteristics. Using the local menu, you can search for an area, zone or owner by number or by name, copy records to and from other applications, and send the records to a printer. You can inspect an area, zone or owner by selecting *Show Dialog* from the local menu, which invokes the [Area Information](#) , [Zone Information](#) or [Owner Information](#) Dialogs. You can also change the format and content of the filters display by selecting the [Display Column/Options](#) item from the local menu. The records can be sorted by any of its component fields simply by clicking the corresponding column's heading.

The display's only enterable field is the one entitled *Shown*, which may assume only the values Yes and No. For example, if the Area/Zone/Owner Filters setting for an area is *No*, then any case information display configured to enforce area/zone/owner filters will omit the area's elements from the resulting record set. You can specify whether a particular case information display enforces filters using the display's [Display/Column Options Dialog](#).

Double-click on a cell in the *Shown* field to toggle its value. Use the [cell handle](#) to propagate a particular value to multiple areas, zones or owners, or use the *Toggle All Yes* or *Toggle All No* local menu options to set the values of all area, zone or owner records.

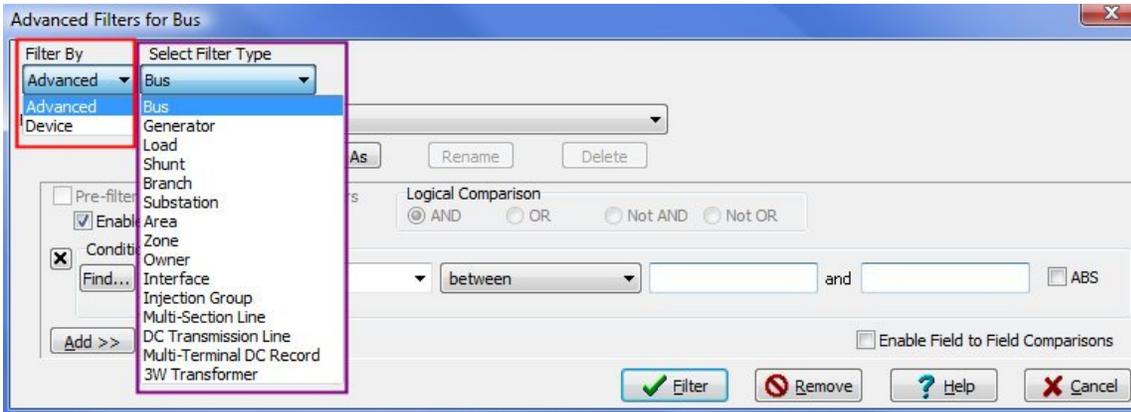
Advanced Filtering

Filtering by areas, zones and owners using [Area/Zone/Owner Filtering](#) is a quick and simple way to filter, however Simulator also contains the ability to perform custom filtering on case information displays as well. Advanced Filtering is accessed by clicking on the [Case Information Toolbar Filtering Menu](#) and choosing Advanced Filter or right-clicking on a [Case Information Display](#) and choosing Advanced Filter from the local menu. This brings up the [Advanced Filters Dialog](#), which allows you to custom filter the information in the display. Advanced Filters are stored with your power system case.

A [Case Information Display](#) will have the phrase "Filter: *Name of Filter*" in its caption if an Advanced Filter has been applied to it. To remove a filter from a [Case Information Display](#), choose Remove Filter from the Case Information Toolbar Filtering Menu or bring up the [Advanced Filters Dialog](#) and click on **Remove**. Note that Remove does NOT delete the filter, but just stops using it to filter the data. You can always remove the filter temporarily and then come back and reapply the filter. If you want to Delete a filter, you must bring it up in the [Advanced Filters Dialog](#) and click on **Delete**.

Advanced Filters Dialog

The first thing to choose on the Advanced Filter dialog is what to **Filter By** and the **Filter Type**.



Filter By

Filter By can be set to either **Advanced** or **Device**.

Advanced Filtering allows you to define a set of logical comparisons for the various fields which must be met. For example you may want to only see Buses that meet the conditions "Nominal Voltage > 100 kV" AND "MW Load > 100 MW". For more help on Advanced Filtering see the topic [Advanced Filtering: Advanced](#).

Device Filtering allows you to directly use one of the power system model objects as a filter. The relationship between the object type being listed and the object type of the device filter determines how filtering is done. For example, if you use an Injection Group as a device filter to filter a list of Generators, then you will get only the generators which are inside the Injection Group. As another example, if you are using an Injection Group as a device filter to filter a list of Branches, then you will get only branches that are connected to the terminal bus of any generator, load, or switched shunt contained in the injection group. Device filtering can be used instead of more complicated Advanced Filters. For more help on Device Filtering see the topic [Advanced Filter: Device](#).

Select Filter Type

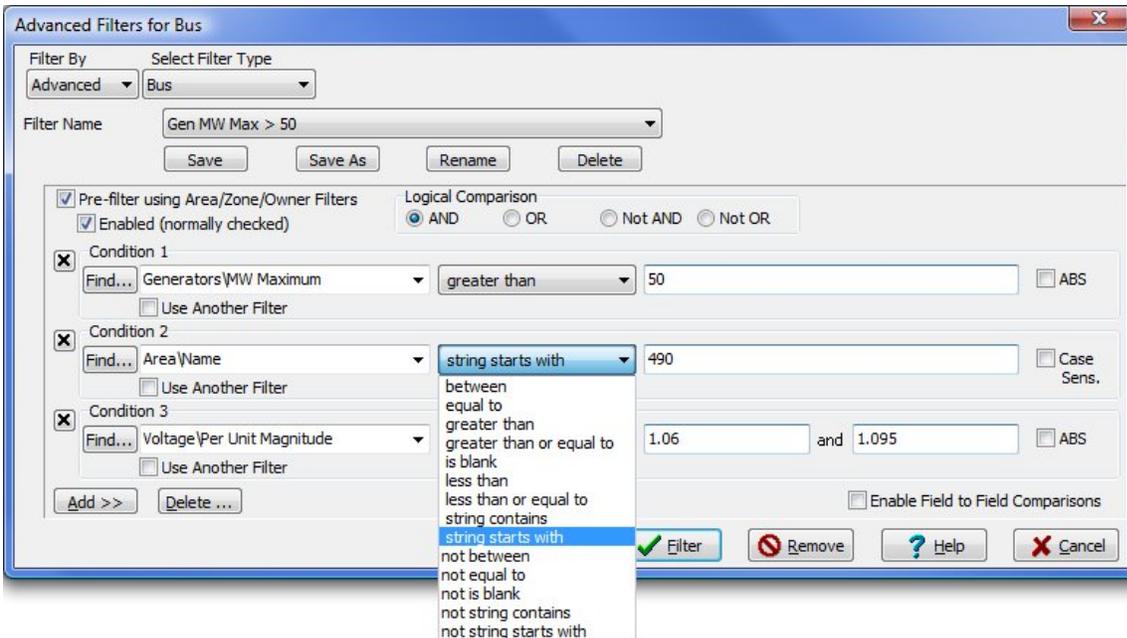
When defining an Advanced Filter, normally this type is the same as the object type for which you are defining a filter. When defining a Device Filter, normally this type will be different.

By changing the drop-down next to Select Filter Type, it is possible to filter based on data for a related object instead of for the actual object. For example, an Advanced Filter for a Bus may exist to only show buses with a Nominal Voltage larger than 138 kV. You may then want to use this filter on a case information display which shows generators. This is possible by changing the Filter Type to be a Bus when showing the Filter Dialog on the Generator case information display. Just click the drop-down to see which related objects' filters can be used for that particular situation.

Note Regarding Auxiliary Files: Throughout auxiliary script commands, the ability to specify an advanced filter by name is available. In this context, if the use of an Advanced Bus filter is desired in a place where an Advanced Generator filter is needed, then use the string "<BUS> filename" to represent the filter. If the use of a device filter is desired, then use the string "<DEVICE> objecttype 'key1' 'key2' 'key3'" to represent the device filter.

Advanced Filtering: Advanced

The [Advanced Filters Dialog](#) **Filter** set to **Advanced** is shown in the following figure.



When you open this dialog, you will only see filters that have been defined for the type of object you are trying to filter (e.g. Bus, Generator, Interface, etc...). You can choose a filter from the Filter Name drop-down box showing the list of filters available, or you can create a new filter. To make a new filter, simply click on Save As to save a copy of the present filter under a new name and then specify the properties of the filter as discussed below. When you have specified the filter as you wish, click **Filter**.

Note that advanced filters are stored with the case file when the case is saved. In addition, the filters can be exported to a Simulator [Auxiliary File](#) for storage and import into other cases. A list of all advanced filters defined for a case can be viewed in the [Advanced Filters case information display](#).

To create an Advanced Filter you must specify the following things:

Filter Name

A string that describes your filter so that you can call it up from other forms in the future.

Condition 1, Condition2, etc...

Describes the conditions of your filter. To define a condition

- Specify the field you are filtering. By default, the fields in the drop-down list are limited to the fields presently shown as columns in the display. If you wish to choose from a list of all fields for the object, change the radio button setting to **Show All Fields**. It is very useful to use the **Find...** button to more easily search for the field you are interested in.
- Specify the comparison operation such as "**between**" or "**greater than**".
- Specify the values the field is compared to. Depending on the comparison operation, either one or two values are needed.
- Select **ABS** to use the absolute value of the field value in the comparison operation. For Fields that are strings, select **Case Sens.** to make the comparison case sensitive.

Note: The comparison operation "within integer range list" uses the same format as described in [Entering a Range of Numbers](#).

Note: To add or delete conditions click on the **Add>>** or **Delete...** button. You can also delete an individual condition by clicking on the **X** button next to the condition.

Note: There is no limit to the number of comparisons you use for the Advanced Filter.

Logical Comparison

The following describes how the Filter uses the Conditions which are specified

- AND means that all conditions should be true
- Not AND is the opposite of AND (i.e. any one of the conditions can be false)
- OR means that any one of the conditions can be true
- Not OR is the opposite of OR (i.e. all of the conditions must be false)

For advanced logical comparisons within the same filter, refer to the important note below.

Pre-Filter using Area/Zone Filters

When this box is checked, data is filtered first by the [Area/Zone/Owner Filters](#) and then by the Advanced Filter, therefore the data must meet both the Area/Zone/Owner Filters and the Advanced Filters in order to be shown. When this box is unchecked, the [Area/Zone/Owner Filters](#) are ignored.

Enable Field to Field Comparisons

When this box is checked, it is possible to compare two fields of the device. In order to do so, select **Field** and click on **Find** to select the second field to be compared. The following depicts what a particular condition looks like when the enable Field to Field comparisons checkbox is checked.

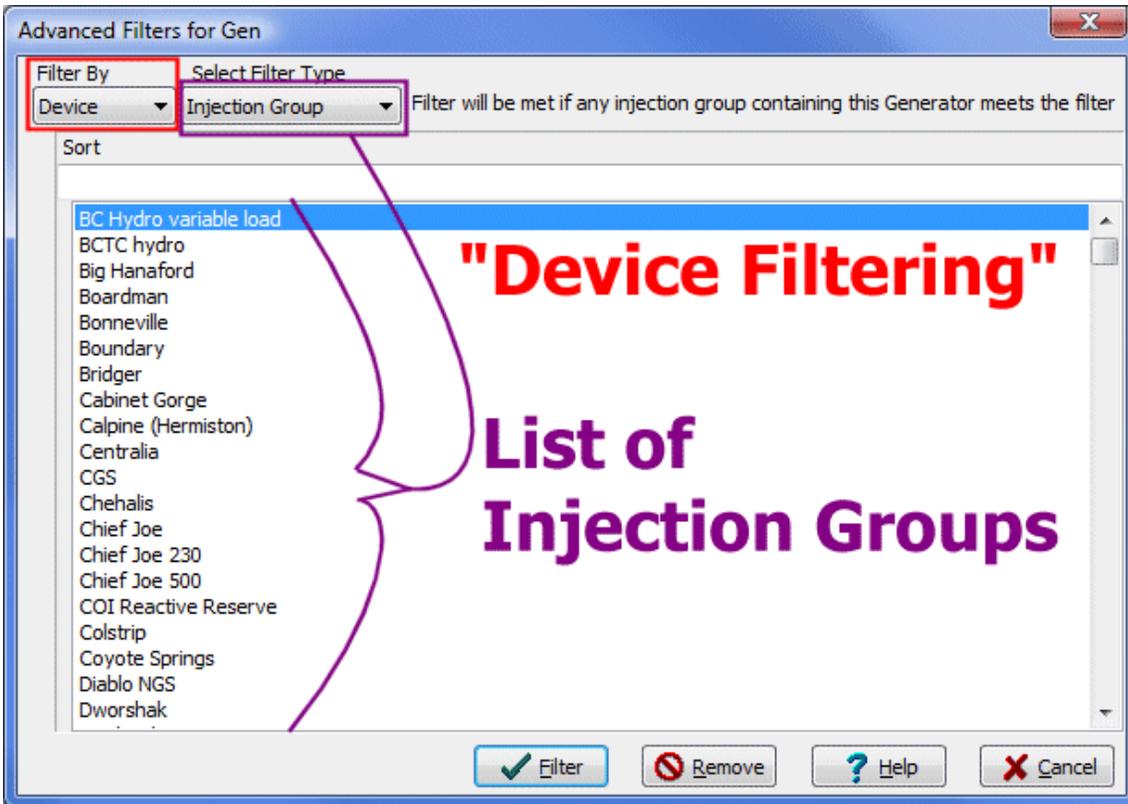
The screenshot shows a configuration window for a filter condition. The title is "Condition 1". It features a "Find..." button and a dropdown menu currently set to "Generators\MW". The comparison operator is "greater than". The second dropdown menu is set to "Generators\MW Maximum". There is an "ABS" checkbox which is unchecked. Below the main configuration, there is a "Use Another Filter" checkbox which is unchecked. At the bottom left are "Add >>" and "Delete ..." buttons. On the right side, there is a "Field" dropdown menu with "Field" selected, a "Find" button, and a "Constant" dropdown menu with "Field" selected. At the bottom right, there is a checked checkbox labeled "Enable Field to Field Comparisons".

Important Note: Once you choose a condition comparison type, that type is used for all conditions in the filter. Therefore if you wish to use the AND condition, all conditions you define will be applied using AND. There is a way to combine different conditions within the same filter. This is accomplished by allowing nested filters. In other words, one condition of a filter can be that another filter is met. To refer to one filter from within another, check the box **Use Another Filter**. Then choose whether the condition is to meet the filter or **Not** meet the filter, then choose the filter (which must have been previously defined) from the drop down list of the third box, or by clicking the Find button. This allows you to define some conditions using one logical comparison in one filter, and then use that filter to combine those conditions with other conditions using a different logical comparison.

For example, consider the logical comparison of **A and (B or C)**. To replicate this, you would define one advanced filter (AF1) that contains the logic **B or C**. Then you can define an advanced filter (AF2) that uses AF1 as **A and (AF1)**.

Advanced Filtering: Device

The [Advanced Filters Dialog](#) **Filter by** set to **Device** is shown in the following figure



Device Filtering allows you to directly use one of the power system model objects as a filter. The relationship between the object type being listed and the object type of the device filter determines how filtering is done. For example, the image below shows the use of an Injection Group as a device filter to filter a list of Generators. The result of using this filter will be only the generators which are inside the Injection Group.

As another example, if you are using an Injection Group as a device filter to filter a list of Branches, then you will get only branches that are connected to the terminal bus of any generator, load, or switched shunt contained in the injection group. Device filtering can be used instead of more complicated Advanced Filters.

Advanced Filters Display

The Advanced Filters Display is a [Case Information Display](#) available under the folder *Case Information and Auxiliary* on the [Model Explorer](#). The purpose of this display is to list any [advanced filters](#) you have defined for the current case. Each record will list the name of the filter, the type of object the filter is for, filter logic, and whether or not a pre-filter was used.

Normally you will create advanced filters directly on a case information display that is showing a particular object type. However you can also choose **Insert** from the [Case Information Toolbar Records Menu](#) to create an advanced filter. You will then be prompted to choose the object type on a dialog before proceeding to the [Advanced Filters Dialog](#).

Normally you will delete advanced filters from the [Advanced Filters Dialog](#), however you can also choose **Delete** from the [Case Information Toolbar Records Menu](#) to delete an advanced filter. This will then remove the filter from any case info displays, [dynamic formatting definitions](#), [select by criteria definitions](#), or any other locations that are presently using this advanced filter.

The most common use of this display is to using the [Case Information Toolbar: Load Auxiliary Files Menu Save As](#) option to save a list of advanced filters to an auxiliary file. feature makes it easy to transfer defined Advanced Filters from one case to another.

Custom Expressions

Simulator allows you to define Custom Expressions that are functions of other fields. These Custom Expressions can then be shown as a column in a Case Information Display. To define custom expressions, right-click in a Case Information Display table and choose **Records > Define Expression** from the local menu or choose **Records > Define Expression** from the [case information toolbar](#). This brings up the Define Custom Expressions Dialog. When you bring up the dialog you will only see expressions that have been defined for the type of object shown on the Case Information Display (e.g. Bus, Generator, Interface, etc...).

To define a new custom Expression click **New**.

You can name the expression for easy identification in the list of fields for the object by filling in the **Expression Name** field.

To define the Custom Expression, first specify which fields you would like to use in the expression and assign them to the variables x1, x2, ..., x8. Fields can be chosen that apply to the type of object for which the expression is being defined or a [Model Expression](#) can be chosen. Model Expressions allow using either an expression that applies to model fields or simply a single model field. After fields or Model Expressions have been assigned to variables, type in the expression as a function of the variables x1, x2, ..., x8. For example

$x1 * \text{SIN}(x2) + \text{EXP}(-x5)$

or

$\text{TAN}(x1) + \text{ABS}(x6)*8 - 100$

For a complete list of functions and operators that are available to you, see [Functions and Operators Available](#).

Once the Custom Expression has been defined, you may add the expression to the column of the Case Information Display from which it was called by clicking on **Add Column...** For more information on how to add columns to a display see [Configuring the Case Information Displays](#).

Custom Expressions Display

To open the Custom Expressions Display, select **Case Information and Auxiliary > Custom Expressions** in the [Model Explorer](#). If you have defined any [custom expressions](#) for your case, you can see the list of those expressions in this display. The type of object to which the expression applies, the name of the expression, and the custom expression itself are given. The display also shows the values being represented by each of the variables in the custom expression.

Custom expressions cannot be inserted manually in this display, but they can be loaded from an [auxiliary file](#) by right-clicking on the grid and selecting **Load > Auxiliary File (any data)...** from the local menu. Conversely, you can save a list of custom expressions in a case to an auxiliary file by right-clicking on the grid and choosing **Save As > Auxiliary file**.

Functions and Operators Available

Below is a list of functions and operators that are available for use in [Defining Custom Expressions](#):

Symbol	Equivalent	Description	Example
()		Prioritizes an expression	$5*(1+1) = 10$
!	FACT	Factorial	$5! = 120$ $\text{fact}(5) = 120$
%		Percentage	$35\% = 0.35$
^	**	Raised to the power of	$4 \wedge 5 = 1024$
*		Multiply by	$3 * 6 = 18$
/		Divide by	$9 / 2 = 4.5$
\	DIV	Integer divide by	$9 \setminus 2 = 4$
MOD		Modulo (remainder)	$7 \text{ mod } 4 = 3$
+		Add	$1 + 1 = 2$
-		Subtract	$9 - 5 = 4$
>		Greater than	$9 > 2 = 1$ * see note
<		Less than	$7 < 4 = 0$
==	=	Equal test	$5 == 4 = 0$
>=	=>	Greater or equal	$3 >= 3 = 1$
<=	=<	Less or equal	$\#h3E <= 9 = 0$
<>		Not equal	$\#b10101 <> 20 = 1$
NOT		Bitwise NOT	$\text{NOT}(15) = -16$
AND	&	Bitwise AND	$\#b101 \text{ AND } \#h1E = 4$
OR		Bitwise OR	$13 \text{ OR } 6 = 15$
XOR		Bitwise Exclusive OR	$9 \text{ XOR } 3 = 10$
EQV		Bitwise Equivalence	$6 \text{ EQV } 9 = -16$
IMP		Bitwise Implication	$1 \text{ IMP } 5 = -1$
IIF		If condition	$\text{Iif}(1+1=2,4,5) = 4$
MIN		Minimum value	$\text{min}(10,3,27,15) = 3$

Case Information Displays

MAX		Maximum value	$\max(1,9)=9$ *see note
SIN		Sine	$\sin(\pi) = 0$ *see note
COS		Cosine	$\cos(\pi) = -1$
TAN		Tangent	$\tan(\pi) = 0$
ASIN		Arc sine	$\text{asin}(1) = 1.570$
ACOS		Arc cosine	$\text{acos}(-1) = 3.141$
ATAN	ATN	Arc tangent	$\text{atan}(0) = 0$
SEC		Secant	$\text{sec}(0) = 1$
CSC		Cosecant	$\text{csc}(1) = 1.18$
COT		Cotangent	$\text{cot}(1) = 0.642$
SINH		Hyperbolic sine	$\text{sinh}(3) = 10.01$
COSH		Hyperbolic cosine	$\text{cosh}(2) = 3.76$
TANH		Hyperbolic tangent	$\text{tanh}(1) = 0.76$
COTH		Hyperbolic cotangent	$\text{coth}(1) = 1.31$
SECH		Hyperbolic secant	$\text{sech}(0) = 1$
CSCH		Hyperbolic cosecant	$\text{csch}(1) = 0.85$
ASINH		Hyperbolic arc sine	$\text{asinh}(2) = 1.44$
ACOSH		Hyperbolic arc cosine	$\text{acosh}(9) = 2.89$
ATANH		Hyperbolic arc tangent	$\text{atanh}(.1) = 0.10$
ACOTH		Hyperbolic arc cotangent	$\text{acoth}(7) = 0.14$
ASECH		Hyperbolic arc secant	$\text{asech}(.3) = 1.87$
ACSCH		Hyperbolic arc cosecant	$\text{acsch}(2) = 0.48$
ABS		Absolute value	$\text{abs}(-8) = 8$
EXP		e to the power of	$\text{exp}(3) = 20.08$
EXP2		2 to the power of	$\text{exp2}(3) = 8$
EXP10		10 to the power of	$\text{exp10}(3) = 1000$
LOG	LN	Natural log	$\log(16) = 2.77$

Case Information Displays

LOG2		Log base 2	$\log_2(8) = 3$
LOG10		Log base 10	$\log_{10}(100) = 2$
CEIL		Round up	$\text{ceil}(6.2) = 7$
RND		Random number	$\text{rnd}(1) = .969$
INT		Truncate to an integer	$\text{int}(6.8) = 6$
SGN	SIGN	Sign of expression (-1, 0, or 1)	$\text{sgn}(-9) = -1$
SQR	SQRT	Square root	$\text{sqr}(64) = 8$

Find Dialog Basics

Many times when working with large load flow cases, it can be somewhat difficult to locate devices in the [case information displays](#) regarding a specific device. Simulator has many tools to facilitate filtering data, such as the [Area/Zone/Owner filters](#) and the [Advanced Filtering](#) tool. Even with these helpful tools, finding a device can still be hampered when the bus number or exact spelling of the bus name are not known.

To facilitate locating devices in Simulator, you can use the Find tool to use Simulator's advanced search engine for finding the device(s) you are looking for. The Find tool is available from the popup menus of almost all [case information displays](#), as well as several of the various information dialogs in Simulator. Anywhere you see a button or menu option labeled **Find...** you can open the advanced search tool.

Once the Find dialog has been opened, the dialog will automatically adjust to suit the type of device you are searching for. The caption of the dialog should reflect the type of object the dialog is currently attuned to locate. For most devices, such as buses, generators, loads, etc., there will be one list displayed at the bottom of the dialog containing numbers and names of the type of device you are searching for. In some instances, mostly when searching for branch-type devices, the bottom panel is split with a second list is displayed in the right side panel. This list is used to display the possible connections of the bus selected in the first list. For example, if bus number one is selected and it has connections to bus two and bus three, the second list will display the information for bus two and bus three. Thus you can search for a bus in the first list, then choose from the possible connections in the second list to get a specific branch from the list.

Despite what type of device you are trying to find, the first few options and buttons on the dialog will be the same. **Sort by Name** and **Sort by Number** allow you to choose how you wish to find a device in the list. If you know the bus number you are looking for, choose **Sort by Number**. If you know the name, or at least part of the name, that you are looking for, then choose **Sort by Name**. The list (or first list for branches) will be sorted accordingly.

If you wish to narrow down the list of devices to search through, you have a couple of options for filtering the list before searching through it. First you can make use of the traditional [Area/Zone/Owner filters](#) by clicking in the associated check box. If you need to set more specific conditions for filtering the list, you can instead click the **Define Filter** button to set up an [advanced filter](#). Either one will reduce the number of devices in the list for the search.

Once you have the list set up for your search, you can type in the number or name you wish to find in the text box. If you do not know the exact number or name, you can use wildcards to facilitate the search and find all possible matches for a set of characters or numbers. You can use a question mark (?) to represent a single character wildcard, or an asterisk (*) to represent a multiple character wildcard.

For example, if you want to find bus number 10005, but all you know is the first four digits are 1000, you can type in *1000*, and Simulator will search until it finds the first number that contains those four digits. You can then use the **Search Next** button (note that pressing **Enter** is the same as clicking **Search Next**) to find the next number containing the four numbers, and so on. The same goes for searching by name. If you are looking for bus ACEONE, but all you know is the name contains the string ACE, then you can type in *ACE* and then keep pressing **Enter** until you find the bus named ACEONE. By using the beginning and ending *, we would also find elements such as NEWACE, because the double * looks for strings that contain ACE anywhere in the string. Note that if you know the first few letters (or numbers), you can narrow down the number of elements found from the search by eliminating the first * from the search string. For example, to find bus ACEONE, we could have instead used ACE*, and this would have gone through all matches that started with ACE, ignoring other elements such as NEWACE. You can also use wildcards in the middle of a string, such as AC*NE, and Simulator will find any name that starts with AC and ends with NE.

Once you have entered a wildcard search, you can also click the **Search All** button. This will essentially bring up the same Find dialog again, but with the choices narrowed to those that meet your wildcard search.

Search for Text Dialog

This dialog allows you to search for specific text in a case information display. Specify the text you want to search in **Search for** edit box. Clicking on **Search Next** will take you to the next field whose content matches with the text specified. The search can be made **By Rows**, in which the text is searched first in all the fields of a record, before searching in the next record. If the search is made **By Columns** then the text is searched first in all the fields of a column, before searching the text in the next column. If the option **Match case** is checked, the search for text will be case-sensitive. The option **Find entire cells only** will take you only to fields whose entire content matches completely with the text you are searching for.

When checking the box **Search Ignore Filters**, Simulator will look through all the records of the type on your present case information display, but will ignore any [Area/Zone/Owner Filtering](#) and any [Advanced Filtering](#) specified with that case information display. If text is found which matches your wildcard search, but is for a record presently filtered out, then you will be prompted to remove the filtering so that the record can be displayed.

When checking the box **Search All Fields**, Simulator will look through every available field which can be displayed for the type of records presently represented on your case information display. If text is found which matches your wildcard search, but is for a field that is not presently shown, then you will be prompted to add the field to the columns that are shown.

Model Conditions Display and Dialog

Model Conditions are a type of Model Criteria. Model Criteria represent Boolean expressions regarding the present state of the power system model. They can be used to create a convenient display that shows whether the power system meets a set of criteria. They can also be used in conjunction with the definition of a contingency in the [Contingency Definition Display](#) to create contingency actions that are conditional based on the state of the power system. To see a list of Model Conditions, choose **Case Information and Auxiliary > Model Conditions**.

A model condition contains two parts: a power system element and an [advanced filter](#). The model condition will then return true or false depending on the result of applying the advanced filter to the power system element specified. Model conditions can also be used as part of a [Model Filter](#).

The Model Conditions Display is a type of [Case Information display](#) and has the abilities common to this type of display. To delete a model condition, right-click on the display and choose **Delete**. To insert a new model condition, right-click on the display and choose **Insert**. This brings up the Model Conditions Dialog that can be used to create, delete, and modify Model Conditions.

Model Conditions Dialog

The Model Conditions Dialog has three sections. The top section provides the ability to **Save**, **Save As**, **Rename** and **Delete** model conditions. To choose a different Model Condition, click on the down arrow next to the Model Condition name.

The middle section provides a location to specify what power system element this Model Condition is related to. On the left is a list of **Element Types** that are available. When clicking on one of these types, the right portion of the dialog will provide you with a list of the elements of this type. This list is a familiar [Find Dialog](#) and provides you the ability to search for the element you are interested in.

The bottom section provides a location to specify an [Advanced Filter](#) which you would like to have applied to the power system element chosen in the middle section. The bottom section of the dialog behaves identically to the [Advanced Filtering Dialog](#). If an Advanced Filter exists for the type of element for which you are creating a Model Condition, you can click the **Set Filter Same As** button to choose this filter. This will then set the parameters of the Model Condition to be the same as the Advanced Filter.

Model Filters Display and Dialog

Model Filters are a type of Model Criteria. Model Criteria represent Boolean expressions regarding the present state of the power system model. They can be used to create a convenient display that shows whether the power system meets a set of criteria. They can also be used in conjunction with the definition of a contingency in the [Contingency Definition Display](#) to create contingency actions that are conditional on the state of the power system. To see a list of Model Filters, choose **Case Information and Auxiliary > Model Filters** in the [Model Explorer](#).

A model filter contains a list of [Model Conditions](#) and a Boolean operator to apply to these model conditions. The model filter then returns true or false depending on the resulting of applying the Boolean operator to the Boolean results of the model conditions.

The Model Filters Display is a type of [Case Information display](#) and has the abilities common to this type of display. To delete a model filter, right-click on the display and choose **Delete**. To insert a new model filter, right-click on the display and choose **Insert**. This brings up the Model Filters Dialog that can be used to create, delete, and modify Model Filters.

Model Filters Dialog

The Model Filters Dialog behaves in a manner very similar to the [Advanced Filtering Dialog](#). To choose model conditions for the model filter, click the down arrows next to the model conditions list. You may also click the **Find...** button to the left of the list. To insert or delete model conditions from the model filter, click the **Add>>** or **Delete** button. When specifying more than one model condition, choose the **Logical Comparison** that will be used to compare the conditions. Finally, the top of the dialog provides the ability to **Save**, **Save As**, **Rename** and **Delete** model filters. To choose a different Model Filter, use click on the down arrow next to the Model Filter name.

To modify the model conditions in the case, click the **Modify Model Conditions Dialog**. This will open the [Model Conditions Dialog](#).

Entering a Range of Numbers

On a number of displays it is often convenient to enter a group of numbers, including ranges. Examples include entering buses or areas to scale on the [Scaling Display](#), or buses on the [Quick Power Flow List](#). The format for this field is to enter individual numbers separated by commas, and/or ranges with a dash between the beginning of the range and the end of the range.

For example the entry

1-5,21,23-25

corresponds to numbers 1 through 5, 21 and 23 through 25.

Copying Simulator Data to and from Other Applications

You may sometimes find it useful to copy data from Simulator to other applications such as a word processor or spreadsheet program. Alternatively, you may want to copy data from other applications into Simulator. Simulator's [case information displays](#) provide a convenient way to accomplish this. In particular, the **Copy All**, **Copy Selection**, and **Paste** options from the local menu or the [Case Information Toolbar: Copy, Paste and Send Menu](#) allow Simulator to pass data with other applications through the windows clipboard.

To copy a selection of data from a case information display to another application, first select the range of cells to copy from the case information display. Then, right-click on the case information display to display its local menu, and choose **Copy Selection**. To copy the entire content of a case information display to another application, follow the same procedure, except choose **Copy All** from the local menu instead of **Copy Selection**. Switch to the application that will serve as destination for the data and use that application's **Paste** command to finish copying the selected Simulator data to that application. Note that not only is the data copied, but by default so are the data headings. The data headings are very important to maintain if you are planning on copying data from a spreadsheet back into a Simulator case information display. Also note that Simulator copies the text to the windows clipboard with tabs delimiting each cell.

Note: Be aware that most programs have limitations on the amount of information you can paste. For example, some spreadsheet programs only allow up to 256 columns and 65,000 rows of information. Some power system information, particularly the Ybus or Jacobian matrices, can easily exceed these limitations.

To paste data from another application into a case-information display, select the data in the other application and use that application's Copy command. In order to paste back into Simulator, you must have the record type and data headings selected with the columns of data. The record type (or object name) must match the case information display you are attempting to paste into, and the data headings (or variable names) tell Simulator which columns of data you are pasting. In order for the data to be pasted to the correct records, you must include the [key field](#) columns in the data to be pasted into Simulator. Also if you are attempting to create new data records by pasting back into Simulator then you must include the [required field](#) columns as well. If the object type is not valid then the **Paste** option will be disabled and the caption will read "object type invalid". If the key fields are missing, then the **Paste** option will be disabled and the caption will read "key fields missing".

It is not necessary to copy and paste all columns or rows of data back into Simulator, as long as the column headings match a valid heading for the case information display. Simulator will skip the data under any column headings that are unrecognized. If you wish to copy data from a spreadsheet into a Simulator display, you must make sure they are ordered in such a way that all the information you wish to paste can be grouped and copied as one block of data from the spreadsheet. Once you have copied all the information from the spreadsheet you wish to transfer, switch to Simulator, open the case information display in which to paste the data, and select **Paste** from its local menu.

Note: You can only paste values into the case information displays if the values are enterable on the display (shown blue by default). Also, be careful about pasting redundant data. For example, in the Bus Records display both voltage in *per unit* and voltage in *kV* are enterable, but they specify the same information. Make sure you only copy ONE of these columns into Simulator. Otherwise you may not get what you expect. Simulator will paste the value in twice, and whatever value was pasted 2nd will be used.

Save Case Information Data

The Save Data Dialog gives you quick access to saving data from certain [Case Information Displays](#) in a PowerWorld Simulator [Auxiliary](#) File.

Specify the name of the auxiliary file in which to save the data records in the text box labeled **Name of File to Save**. Instead of typing the name of the file by hand, you can press the **Browse** button to locate it. Then, indicate whether you want the objects whose data you are writing to the file to be identified by number or by name. Finally, to ensure that only the records currently listed in the [Case Information Display](#) are written to the auxiliary file, check the **Save Only Records Listed in this Display** checkbox. Otherwise, data for all such objects in the entire system will be saved to the file.

When you have finished setting these options, click **OK**. If you changed your mind and do not want to save the data to a file, click **Cancel**.

Key Fields

Key fields are necessary fields when attempting to paste data into a [Case Information](#) display from another data application. Other fields are also considered [required fields](#) for the purpose of creating new objects in Simulator by pasting or loading information from another source, such as Excel or a [PowerWorld Auxiliary](#) file. The key fields are easily identified from a few different locations.

Column Headings

The column headings of case information displays are colored to indicate fields that are [key fields](#). Any field that is a [key fields](#) will be highlighted yellow. Also note that some fields are highlighted green. These fields are [required fields](#) that are necessary when you are attempting to create a new object by pasting or loading the data from another application. If the necessary [key fields](#) (yellow) are not included in the data loaded from another source, the information for that object is ignored. If all of the [required fields](#) (green) are not included in an object that is detected as new, Simulator will skip it. If all the required fields are present, then Simulator can create the new object. Note that for new objects, the key fields are necessary as well.

Display/Column Options

The Display/Column Options dialog, available by right-clicking on any case information display, also has the capability to highlight both the [key fields](#) and [required fields](#), as described above. The check box labeled "Highlight Key Fields" will enable highlighting of the field names in the two lists on the display.

Export Object Fields

This option from the Help menu in Simulator allows you to export a list of most fields for each type of object in a case. The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.

The following table lists some of the more commonly accessed key fields needed when attempting to Paste data into existing objects in a Case Information Displays from another data application.

Type of Data	Data Description	Necessary Key Columns for Paste		
Area	Area Records	Area Num		
Bus	Bus Records	Number		
DC Line	DC Line Records	Rectifier Number	Inv Number	Num
Gen	Gen Records	Number	ID	
Interface	Interface Records	Name		
Line/Transformers	Line Records	From Number	To Number	Circuit
Load	Load Records	Number	ID	
Substations	Substation Records	Sub Num		

Case Information Displays

Switched Shunt	Switched	Number	ID
	Shunt Records		
Zone	Zone Records	Zone Num	

Required Fields

Required fields are necessary fields when attempting to create new objects by pasting data into a [Case Information Display](#) or when reading data from an [PowerWorld Auxiliary](#). These fields are in addition to the [key fields](#) which identify the object.

There are two purposes of the required fields. First there are some fields for which an appropriate default value is not apparent thus by requiring these fields it forces the specification of the value. Second, when pasting data into Simulator it is possible that the record will not exist in the case. Simulator will then determine whether to create these records by looking to see if the required fields are specified. By doing this, it prevents the accidental addition of records. For instance you may have an Auxiliary File that sets the Monitor field of all branches in your case to Yes or No. If Simulator runs across a branch between two buses that exist, but whose circuit ID does not exist, then Simulator will just skip this line from the Auxiliary File. It will not be possible to accidentally create a bunch of new transmission lines with default impedance parameters because the R and X impedance parameters are required.

The required and key fields are easily identified from a few different locations.

Column Headings

The column headings of case information displays are colored to indicate fields that are [key fields](#). Any field that is a [key fields](#) will be highlighted yellow. Also note that some fields are highlighted green. These fields are [required fields](#) that are necessary when you are attempting to create a new object by pasting or loading the data from another application. If the necessary [key fields](#) (yellow) are not included in the data loaded from another source, the information for that object is ignored. If all of the [required fields](#) (green) are not included in an object that is detected as new, Simulator will skip it. If all the required fields are present, then Simulator can create the new object. Note that for new objects, the key fields are necessary as well.

Display/Column Options

The Display/Column Options dialog, available by right-clicking on any case information display, also has the capability to highlight both the [key fields](#) and [required fields](#), as described above. The check box labeled "Highlight Key Fields" will enable highlighting of the field names in the two lists on the display.

Export Object Fields

This option from the Help menu in Simulator allows you to export a list of most fields for each type of object in a case. The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.

An example of required fields for an object is for a Branch object. For a branch object, the key fields are the from bus number, to bus number and the circuit ID. Required fields are the Impedance Values R, X, and B and the first three ratings A, B, and C. Without specifying values for these fields you can not create a new branch.

Contour Column Dialog

The Contour Column Dialog allows you to contour case information display columns.

The Contour Column Dialog has two tabs:

[Contour Type](#)

[Custom Color Map](#)

Contour Column Type

Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the [Custom Color Map](#) Tab.

Reverse Color Map Colors

Check this box to reverse the colors of the selected color map, so the low color becomes the high color, and vice versa.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Use cell values directly

Select this option to use the cell values to do the contouring.

Use the specified field below

Select this option to use a different field value to do the contouring.

Value

Select the quantity to use in the contouring from the Value dropdown box or click the **Find Value** button to find the desired field.

Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values. You can also give the color key a title, unit label, and specify the number of digits to display in numerical values.

Title

Title for the color key.

Entry Labels

Units of the contoured value displayed on the color key.

Dec. Pts.

Number of decimal places of the contoured value displayed on the color key.

Scalar

Multiplication factor that can be applied to the values when drawing the color key.

Use Equal Spacing For Discrete Maps

This option will draw the color key with equal spacing for all colors in the map, regardless of how close or distant the values the colors represent.

Use absolute value

Check this check-box to use the absolute values of the quantity selected at the Value dropdown box (above).

Values

These values along with the color map define how to convert your values into a color for the contour. The values are:

Maximum The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Break High This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

Nominal This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

Break Low This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

Minimum The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Note: a representation of the color map is shown to the right of the values.

Grid Metrics Dialog

The grid metrics dialog allows you to determine the metrics for a set of columns. The metrics determined include: Sum, Average, Variance, Standard Deviation, Maximum, Minimum, Total Items, and Total Non-Zero Items.

By default the dialog is set to determine the metrics for the whole column of the case information display where the option has been selected. However, the selected record set can be modified. The **Start Column** and **End Column** values specify what columns to include in determining the metrics. The **Start Row** and **End Row** values specify what records to include when computing the metrics. If the option **Treat blank cells as zero** is checked, then any blank cells will be considered as zero when computing the metrics. Instead, if the option **Ignore blank cells for calculation** is checked, any blank cell will be taken out of the metrics calculation. Check **Use Absolute Values** to use the absolute values of field values for the metrics computation. Finally, click **Update Metrics** to determine the metrics with the new set of options.

Auxiliary File Export Format Description (For both Display and Power System)

Definition of an Export Format Description

An export format description consists of the following items:

1. A set of object types
2. The fields and subdata to be exported for each object type, including how each field should be formatted when written (e.g., number of digits, number of decimal places)
3. The format (comma-delimited, space-delimited, etc.) that should be used when writing the object data to disk.

For instance, an export format description might have two object types defined—buses and generators—with indications to export the bus numbers for each of these objects and to write this information to a space-delimited .AUX file.

Types of Export Format Descriptions

Export format descriptions can be defined for both the Display Objects and the Power System Objects,

- Display objects, e.g., background lines, pie charts, displayed fields, etc. Export format descriptions for display objects are defined by selecting Display Objects Export Format Description... on the [Onelines](#) ribbon tab, under the List Display Menu on the **Active** ribbon group.
- Power system objects, e.g., buses, loads, generators, etc. Export format descriptions for power system objects are defined by selecting Power System Objects Export Format Description from the [Case Information](#) ribbon tab on the **Case Data** ribbon group.

The dialogs used to define both types of export format description are very similar; any differences between the two dialogs will be clearly noted below.

Export Format Description Dialog

Clicking on **Display Objects Export Format Description** or **Power System Objects Export Format Description**, opens the Export Format Description Dialog. There are several buttons at the bottom of the dialog:

- **Create Format for Complete Case** : sets the format for the AUX file to be created to be all necessary fields to replicate the load flow case. This is essentially the same as choosing to save the case as an auxiliary file when choosing "Save Case As" from the Application menu and setting the "Save as type" parameter to "PowerWorld Auxiliary Data".
- **OK** : save the current settings to the selected export format description, and close the dialog
- **Create AUX File with Specified Format** : writes data to a file using the current settings. A dialog pops up to select the filename for storing the data. If the file already exists, the data is appended; otherwise, a new file will be created
- **Cancel** : do not save the current settings and close the dialog
- **Help** : open the help associated with the dialog .

The other parts of this dialog box are discussed in detail below

Format Name Section

The uppermost part of the dialog box is used to manage the set of export descriptions. Once one export description has been saved (by clicking Save or Save As, after defining some object types), a drop-down box will appear which allows selection of an export format description from the set of export format descriptions saved with the case: The buttons underneath the drop-down box have the following functions:

- **New** : clear the rest of the form in order to define a new export format description
- **Save** : save the information in the rest of the dialog box to the export format description currently selected in the drop-down box. If no export format description is selected, this is equivalent to clicking Save As.
- **Save As** : save the information in the rest of the dialog box to a new export format description; a dialog will pop-up asking for the name of the new export format description.
- **Rename** : rename the currently selected export format description
- **Delete** : delete the currently selected export format description

The list of export format descriptions for display objects and power system objects is kept separate, so the drop-down box will only contain export format descriptions defined for the object type selected from the menu (Display Objects Export Format Description Dialog or Power System Objects Export Format Description Dialog).

File Type and Field Defaults

The next section of the dialog box is used to define the format to save the data, and the default formatting of any numeric fields that are to be saved. Currently, there are 3 options for the file type:

- **AUX (space)** : output the data using a space-delimited .aux file format
- **AUX (,)** : output the data using a comma-delimited .aux file format
- **CSV** : output the data to a comma-separated-values (CSV) file. This type of file can be easily imported into any spreadsheet or database software.

In the **Defaults for Exporting Fields** section, the up/down arrows can be used to change the default number of total digits and digits to the right of the decimal place used in exporting numeric values. These values may be overridden in the Select Fields form for individual fields.

Object Type and Filter Method Section

The two-column section on the left side of the dialog box is used to define object types and a filter for each object type.

Object Type

The first column is used to specify which object types are to be exported. To insert a new object type (corresponding to a new row) into the export format description, there are two options—either click the New button above the **Object Type** column heading, or right click on a row and select **Insert**. Upon doing this, a dialog will pop up to choose an object type:

Type into the text box to search for a particular object type. The list of object types depends on the type of export description being defined. When defining display object export descriptions, only display object types will show up in this list; similarly, when defining power system export descriptions, only power system objects will show up.

Clicking **Choose** will insert the object type into the list of object types to be exported.

Double-clicking an existing object type will pop-up this same dialog box, allowing the object type to be changed. Doing so will empty the exported fields, subdata, and filter method.

Filter Method

The second column is used to define an object filter. The default behavior is to export all objects without any filtering, but this can be changed by double-clicking on the filter description. Upon double-clicking on the filter description, a dialog will appear allowing you to choose the filter. The options available for determining which objects are exported are as follows

- **All** : export all objects, disregarding any filters defined in the current case
- **Use Area/Zone/Owner Filter** : only export those objects that satisfy the currently defined area/zone/owner filter. Clicking on **Select Area/Zone** will pop up the dialog box used to define area/zone/owner filters.
- **Use Selected** : only export those objects that have the Selected field set to Yes. Clicking on **Select** pop up the dialog box used to set the Selected flag for each of the objects in the case. Note that this particular option does not exist for display objects, because display objects do not currently support the Selected field.
- **Meets Filter** : only export those objects that meet an advanced filter, which can be either selected from the drop-down box or defined using the **Define Filter** button.

Fields And Subdata To Be Exported

After specifying an object type (and, optionally, a filter), selecting the row on the left side of the dialog box will allow you to specify the fields and subdata to be exported.

Fields To Be Exported

By default, no fields are selected for export. To define fields for export, click on the **Modify** button to the right of the Fields heading the **Select Fields dialog** box will popup. This dialog is used to specify which fields are to be exported, along with any field-specific formatting of numerical values . Generally the dialog behaves identically to the [Configuring Case Information Displays Dialog](#), so for more detailed help see that topic. There are parts of the dialog that behave differently however, and these are described as follows.

- **Total Digits / Dec Places** : these two boxes are used to set the number of total digits and the number of decimal places to use when writing the data for the fields current selected in the Selected Fields section. Leaving these two boxes blank means that the default number of digits and decimal places should be used. Also, if multiple fields are selected that have different settings for the number of digits or number of decimal places, these boxes will be grayed out, but still enterable. Once entering a value into either box, all selected fields will then have the same setting, and the box will no longer be greyed out.

Subdata To Be Exported

By default, no subdata is selected to be exported. Also, many objects (e.g., Bus and Area) do not have subdata associated with them, so the **Modify** button will be greyed out. If an object type does have subdata, then the SubData definition section will be grayed out.

To add subdata that should be exported, click the **Modify** button to the right of the SubData heading. This brings up the Select SubData dialog. A description of the options/settings on this dialog follows.

- **Available SubData** : lists the SubData that exist for this object type. If Exclude subdata already selected is checked, then only SubData that is not already in the Selected SubData section will show up in this list.
- **Exclude subdata already selected** : if checked, then the Available SubData section will only contain items that are not already in the Selected SubData section
- **Selected SubData** : lists the SubData to be exported for this object type
- **Add ->** : set the currently selected SubData in the Available SubData section to be exported
- **<- Remove** : set the currently selected SubData in the Selected SubData section to not be exported
- **OK** : save the settings and close the dialog
- **Save** : save the settings, but leave the dialog open
- **Cancel** : do not save the settings and close the dialog
- **Help** : click on this to open the help associated with the Select Fields dialog

Geographic Data View

Geographic Data Views allow the quick creation and formatting of graphical representations of buses, generators, loads, switched shunts, transmission lines, substations, areas, zones, super areas, and injection groups. Latitude and longitude coordinates specified with buses and substations are used to place objects geographically on a display. The latitude and longitude coordinates must be specified in order to use this feature. Object attributes such as color, size, rotation rate, rotation angle, and visibility can easily be formatted based on the associated data object field values.

To create a Geographic Data View, open a case information display for the desired element type, select the elements for which to add geographic representations, and either right-click on the object grid and choose **Geographic Data View > Geographic Data View** from the local menu or choose **Geo > Geographic Data View** from the [case information toolbar](#). Use the **Select Column, then Geographic Data View** option to select an entire column of elements. The column in which the elements are selected determines the initial selection of the field to use for [attribute settings](#). The Geographic Data View Customization dialog will open.

Geographic Data View Customization Dialog

Online for Geographic Data Addition

When creating a new geographic data view, the online to which to add the geographic data view objects must be set. Choose either to **Use Already Open Online** and then select a online from

the dropdown list of available onelines or choose to **Open a Online** and then either type in the name of a PWD file or click Browse to select a one.

Because geographic data views are dependent on having geographic latitude and longitude information available, the online that is selected must have a valid map projection in use. Only those open onelines that currently have a [map projection](#) in use will be listed in the dropdown box and any online opened from a file will be checked for a valid map projection. If a valid map projection does not exist, new geographic data view objects will not be added to the online.

Create Online with Geographic Borders

Click this button to create a new online diagram with first being prompted to add geographic borders. A default name will be assigned to the new online and that online will be added as the selected online in the Use Already Open Online dropdown.

Setting Attributes for Geographic Data View Display Objects

Before adding new geographic data view objects, the attributes associated with the objects should be set. These attributes are stored in [Geographic Data View Styles](#).

Create Empty Online

Click this button to create a new online diagram. A default name will be assigned to the new online and that online will be added as the selected online in the Use Already Open Online dropdown.

Create Geo Data View Display and Close Dialog

Click this button to accept all of the option settings, add new geographic data view objects to the selected online, and then close the dialog.

Cancel

Click this button to close the dialog without adding and new geographic data view objects.

Geographic Data View Styles: Fields and Attributes

The Field and Attributes options associated with Geographic Data View Styles allow the formatting of the geographic data view display objects depending on the values of fields of their linked data objects.

Attribute

Most of the attributes can have a **Default** value or a value that is dependent on the value of a selected field. When the **Use** box is checked for a particular attribute, the field selected in the **Field Lookup** is used to determine the value of the attribute based on either a Color Map or Value Map lookup. Otherwise, the default value will be used. Line Thickness, Total Area, Rotation Angle, and Rotation Rate attributes all use value map lookups. Line Color and Fill Color both use color map lookups. When determining the Field Lookup, either select the field from the drop-down or use the Find button to open the [Find field dialog](#) for easy searching.

When an attribute is selected, either by clicking on one of the options settings for that attribute or just clicking the background box for that attribute, the appropriate color map or value map for that attribute will be displayed in the right-hand panel.

Line Thickness

Thickness of the line making up the outside border of the display object.

Line Color

Color of the line making up the outside border of the display object.

Fill Color

Fill color of the display object. The Fill Color will be used only if the **Use Fill** box is checked.

There are two choices of Field Lookup for this attribute. The top choice is enabled regardless of the choice of object [Style](#). The bottom choice is only enabled when the Kite style is chosen. The Kite style allows two different fields to be displayed in one object. This is useful for displaying

fields like Mvar Maximum and Mvar Minimum. The top field indicates what field will be used to format the top half of the Kite, and the bottom field indicates what field will be used to format the bottom half of the Kite.

When using the Kite style, use of the two different Field Lookups can be selected independently and each has a separate color map. When not choosing to use one or both of the Field Lookups, that appropriate half of the Kite shape will use the default Fill Color choice. To switch between color maps for the two Field Lookup choices, either click on the field drop-down for the

appropriate half of the Kite or click the  button that will appear next to the field choices when the Kite style is selected.

Total Area

This determines the size of the display object. Check the option to **Use Auto Width of Objects** for the width of the objects to be determined automatically. If this option is not checked, the **Width** can be set manually. The height or size is always determined as a function of the width and can be set with the **Size to Width Ratio**. When using the Field Lookup table the total area of the object is determined based on the Characteristic Value returned. The dimensions of the object are then determined based on the total area and the Size to Width Ratio.

There are two choices of Field Lookup for this attribute. The top choice is enabled regardless of the choice of object [Style](#). The bottom choice is only enabled when the Kite style is chosen. The Kite style allows two different fields to be displayed in one object. This is useful for displaying fields like Mvar Maximum and Mvar Minimum. The top field indicates what field will be used to format the top half of the Kite, and the bottom field indicates what field will be used to format the bottom half of the Kite. Even though two fields can be chosen when using the Kite style, only one value map is used for the Field Lookup. When using the Kite style and using the Field Lookup the total area of the object is based on the sum of the lookup values returned for both fields.

Rotation Angle

Static angle of rotation of a display object in degrees.

Rotation Rate

This value is given in Hz and indicates how many times per second an object should do a full rotation. This is used in conjunction with the [General Option for Animation Control](#). If animation control is not enabled, the rotation rate has no effect on how the objects are displayed.

Visibility

This attribute can prevent display objects from being shown if certain conditions are not met. Choosing the **All Visible** option will always show all of the display objects associated with this style. If choosing the option **Only visible if field is non-zero**, select a field from the drop-down box. Only those objects whose selected field is non-zero will be visible. If choosing the option **Only visible if filter is met**, select a filter from the drop-down box or click the Find button to search for an existing filter or create a new one. Only those objects that meet the selected filter will be visible. Note that the visibility attribute is only applied while in Run Mode. All objects will be visible in Edit Mode regardless of how this option is set.

Color Map

Color maps are used with Line Color and Fill Color attributes. The Field Set Metrics section simply gives some statistics on the values of the selected field for the data objects. These metrics are provided to aid in the setting of the breakpoint values for the color map.

Color Map

Choose from the predefined color maps using the drop-down.

Reverse Color Map Colors

This checkbox reverses the mapping of the color map, converting the colors corresponding to the high values into the colors for the low values, and vice-versa.

Use Discrete Color Map

Check this box to make a discrete color map, that is without having smooth transitions between colors.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Use Absolute Value

Check this box to use the absolute values of the field values when mapping the field values to colors.

Breakpoint Values

The breakpoint values determine how the field values will map to colors.

Set Breakpoints from Metrics

As different fields are chosen for attributes, the Field Set Metrics will be updated to reflect the field values. Click this button to update the breakpoint values based on a new set of Field Set Metrics.

Value Map

Value maps are used with Line Thickness, Total Area, Rotation Rate, and Rotation Angle attributes. The Field Set Metrics section simply gives some statistics on the values of the selected field for the data objects. These metrics are provided to aid in the setting of the Field Values and corresponding Characteristic Values.

The attribute value for a particular display object is set to the Characteristic Value that corresponds to the field value of the corresponding data object. Characteristic values will be interpolated if the field value does not fall exactly on a value that has been defined in the table. To insert entries into the table, right-click on the table grid and choose Insert from the local menu. Choose Delete from the local menu to remove entries.

Line Thickness Value Lookup

Field Set Metrics

Min	1.0000	Median	4.0000
Max	7.0000	Average	4.0000

Field Value	Characteristic Value
1.000000	1
4.000000	2
7.000000	3

Attributes for All Objects In Style

These attributes apply to all objects independent of any specific field value.

Layer

Type in the name of an existing layer to assign the objects to an existing layer, or type in the name of a new layer to have that layer created and add the objects to that layer.

Stack Level

An object's stack level dictates what objects it will appear above and which objects it will appear below on a oneline display. For example, circuit breaker and pie chart objects have a default stack level of Top. Therefore, anything with a stack level of Middle, Background, or Base will appear underneath pie charts and circuit breakers on the oneline display. Objects that are within the same stack level and are drawn in the same location will result in the last object drawn being the visible object on the display.

Line Dashed

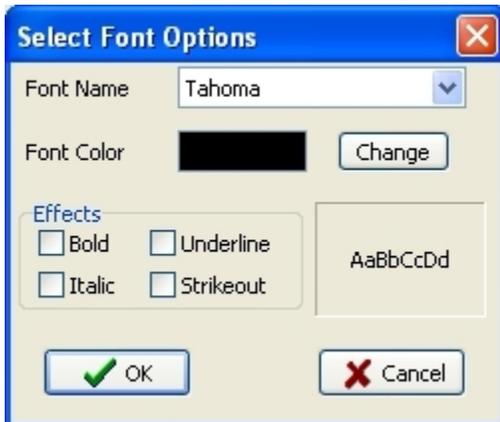
This determines the appearance of the line making up the outside border of the display object. This line can be solid or one of the dashed options.

Immobile

Check this box to keep the geographic display objects from being moved on the oneline diagram.

Set Font

Click this button to bring up the Select Font Options dialog. The changes to font are limited to Font Name, Font Color, and special Font Effects of Bold, Underline, Italic, and Strikeout. There is no option for setting the font size because this is automatically determined based on the size of the display object.



Geographic Data View Styles: General Display Options

The General Display Options associated with Geographic Data View Styles are applied to all geographic data view objects independent of any field value.

Style

The style determines the shape of the geographic data view display object used to represent the selected data records.

Object ID

Each geographic data view display object can be identified by name, number, or a combination of the two. When a choice other than *None* is selected, the appropriate identification for that object will be displayed. When **Prefix ID with Object Type** is selected, the object type will be prefixed along with the main identifier. This option is useful when trying to distinguish between the different types of objects shown on a given display.

Selection Options

Geographic data view display objects can be formatted differently when they are selected as opposed to not selected. Check the option **Can Be Selected** to allow objects to be formatted based on the set options when they are selected. **Line Thickness** determines the thickness of the border line around the outside of the display object when it is selected. **Line Color** determines the color of the border line around the outside of the display object when it is selected.

Animation Control

Geographic data view display objects are capable of being animated if a [Rotation Rate](#) other than zero is specified with the [attributes](#). Check the **Enable Animation** box to animate the objects using this style. The **Interval** determines how often the display is refreshed during animation.

Geographic Data View Styles

Each geographic data view object has an associated Geographic Data View Style. The style determines how the object is going to be displayed and contains options for setting various attributes associated with the object. Each style refers to a particular type of object. Geographic data view objects cannot be individually formatted; they will always obey the formatting of their associated style.

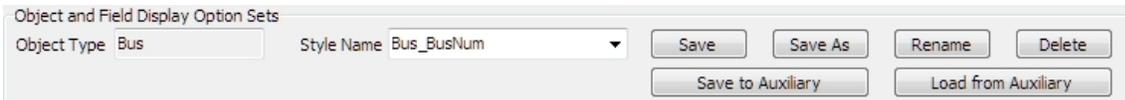
The **Geographic Data View Options** dialog is used to create and modify styles that will determine the appearance of geographic data view objects. This dialog is part of the [Geographic Data View Customization](#) dialog when adding new geographic data view objects to a display, but it becomes a stand-alone dialog when modifying the styles for existing objects. To open as a stand-alone dialog, right-click on an existing geographic data view object and choose **View Geographic Data View Options** from the local menu.

Usually, groups of objects share the same geographic data view style. Keep in mind that changing the options in a style will change all of the objects that use this style.

Geographic Data View Options Dialog

The Geographic Data View Options dialog is made up of four main sections and tabs:

Object and Field Display Option Sets



This section allows the saving, renaming, and deleting of Geographic Data View Styles. Only those styles for a particular **Object Type** will be displayed in the **Style Name** dropdown box. Style names must be unique within an Object Type. Geographic data view styles can be stored in the standard [auxiliary file](#) format. When saving or loading an auxiliary file from this dialog, all geographic data view styles will be stored or loaded regardless of the current object type. The default is to store the information in a data auxiliary file, .AUX, but these objects are also supported in display auxiliary files, .AXD.

[General Display Options](#)

[Fields and Attributes](#)

Object Records

The object records tab lists the data objects that are linked to the geographic data view style, through their associated geographic data view display objects, selected when the dialog is opened. The caption of this tab will be updated to reflect the class type of object that is represented. For example, if viewing bus objects, the caption will be *Bus Records* instead of *Object Records*. These are either the data objects selected before creating a new geographic data view or data objects represented by existing geographic data view objects that are using the selected style. The format of the display objects represented by these data objects will be changed based on any changes made to the options. To assign a group of geographic data view display objects to a new style, simply select a different style than the style in place when opening the dialog and click either the **OK** button on the **Geographic Data View Options** dialog (existing data object was selected) or the **Create Geo Data View Display and Close Dialog** button on the **Geographic Data View Customizations** dialog (new geographic data view objects are being created).

To change the style of a single geographic data view display object and assign it to a new style, use the Display Explorer. Within the Display Explorer, [case information displays](#) for geographic data view objects are available. The style is given in the field **Geographic Data View Style Name** (variable

name GEODATAVIEWSTYLENAME). To change the style, change the name listed in this field to a style that is available for the type of object being changed.

Custom Case Information Display

To open the Custom Case Information Displays, go to the [Case Information](#) ribbon tab, and choose **Custom Case Info** from the **Case Data** ribbon group

Custom Case Information Displays can be used to create a display very similar in appearance to a spreadsheet workbook with several worksheets. This display can show any information that can be shown on a [case information display](#), but also allows you to customize the layout of the information in any manner.

There are three buttons which allow you to **Rename** the present sheet, add a **New** sheet, or **Delete** the present sheet.

There are three different kinds of cells allowed on this display

- Blank Cell – contains nothing
- Plain Text Cell – contains a user-entered string with no link to any data in the model.
- Model Field Cell – contains a link to model field similar to inserting a [model field](#) on a oneline diagram

Each cell of the display will behave differently depending on the Custom Case Info Mode. There are four distinct modes that control the user interaction and operation of the custom case information display. These four modes and the effect they have on the display are described in topics

- [Define Fields/Strings](#)
- [Change Field Data](#)
- [Show Fields Primary](#)
- [Show Fields Secondary](#)

Define Fields/Strings

The "Define Fields/String" mode is the primary mode for use when setting up a new [Custom Case Information Display](#). The three kinds of cells behave as follows in this mode.

Blank Cell

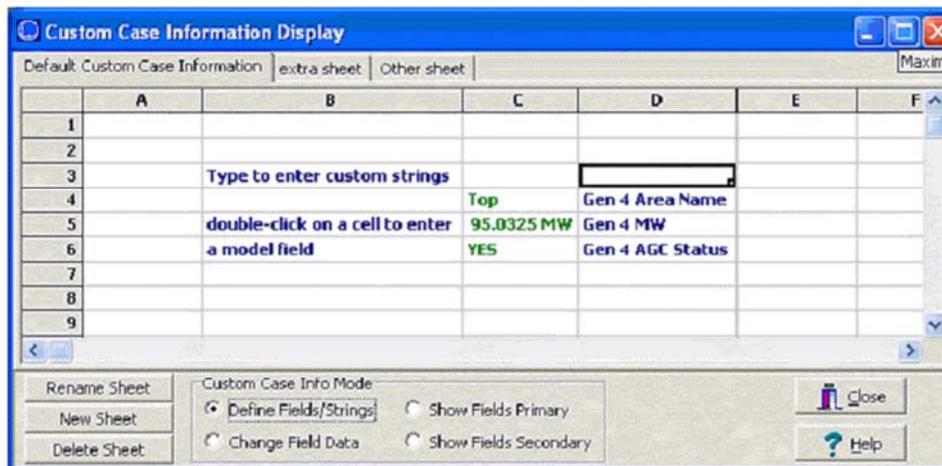
To convert to a Plain Text Cell, just type in the cell. To convert to a Model Field Cell, double-click on the cell to open up a dialog for defining the model field.

Plain Text Cell

These cells will appear in the Case Info Display Enterable Color (blue by default). To change them just type on the cell. When pasting into such a cell or editing it directly it will parse the string entered trying to create a Model Field as though the string represents the model fields as shown in the Show Fields modes. If the string does not represent such a field, then it will remain a Plain Text Cell

Model Field Cell

Model Field cells will not be enterable. In order to edit these fields, you must double-click on the cell to open a dialog for defining the model field.



Change Field Data

The "Change Field Data" mode can be used to edit the data referred to by the model fields on a custom case information display. The three kinds of cells behave as follows in this mode.

Blank Cell

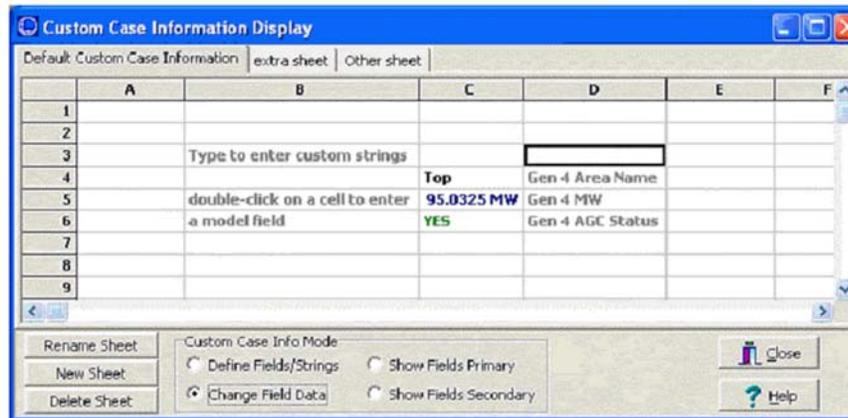
Blank cells may not be edited in any manner.

Plain Text Cell

Plain Text cells may not be edited in any manner. It will appear in a special color defined for the custom case information display which may be specified from the [local menu](#) of the workbook tabs. By default this color is dark gray.

Model Field Cell

Model Field cells will behave according to the field to which they refer. Enterable fields will be enterable, toggleable fields will toggleable, etc... When pasting in the sheet in this mode you will be modifying the model data directly. In the following picture there are three fields in cells C4, C5, and C6 that refer to read-only, enterable, and toggleable field respectively. The fields are labeled using the plain text cells in D4, D5, and D6.



Show Fields Primary

The "Show Fields Primary" mode is used for interacting with an external spreadsheet to create the model field links in custom case information displays. The three kinds of cells behave as follows in this mode.

Blank Cell

Blank cells behave the same as for the "[Define Fields/Strings](#) " mode.

Plain Text Cell

Plain Text cells behave the same as for the "[Define Fields/Strings](#) " mode.

Model Field Cell

Model field cells will display a string which represents information about the model field link. The format for this string will be

```
`model field name' `variable name' totaldigits decimalpoints IncludeUnits
```

The model field string will use the primary [key fields](#).

When pasting into such a cell or editing it directly it will parse the string entered trying to create a Model Field as though the string represents the model fields as shown in the Show Fields modes. If the string does not represent such a field, then it will remain a Plain Text Cell.

Case Information Displays

Custom Case Information Display

Records ▾ Set ▾ Columns ▾ Options ▾

Default: Custom Case Information

	A	B	C	D	E
1					
2					
3		Type to enter custom strings			
4			'Gen<KEY1>2<KEY1><KEY2>1<KEY2>' AreaName:1 8 4 YES	Gen 1 Area Name	
5		double-click on a cell to enter a model field	'Gen<KEY1>1<KEY1><KEY2>1<KEY2>' GenMW 8 4 YES	Gen 1 MW	
6			'Gen<KEY1>1<KEY1><KEY2>1<KEY2>' GenAGCable 8 4 YES	Gen 1 AGC Status	
7					
8					
9					
10					
11					
12					
13					
14					
15					
16					
17					
18					
19					
20					
21					
22					
23					
24					
25					
26					
27					

Rename Sheet Custom Case Info Mode Close

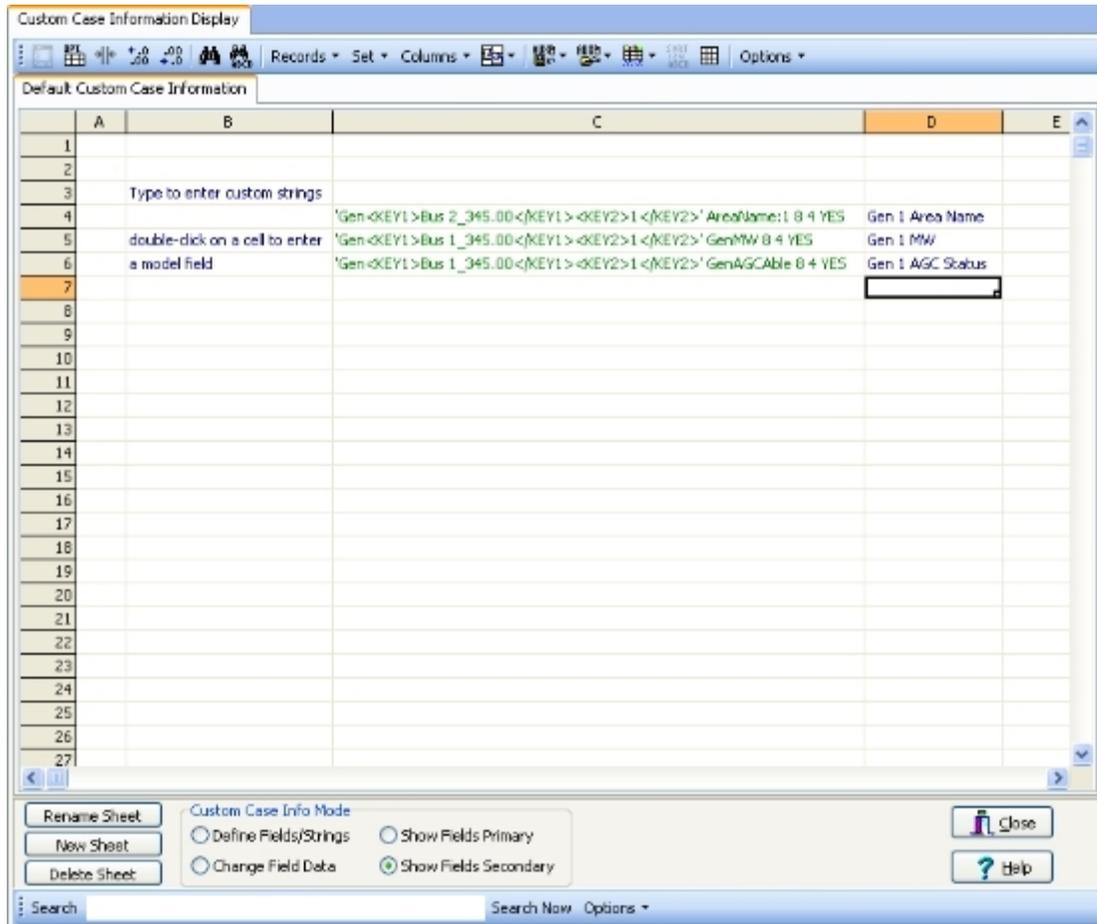
New Sheet Define Fields/Strings Show Fields Primary

Delete Sheet Change Field Data Show Fields Secondary Help

Search Search Now Options ▾

Show Fields Secondary

This mode is identical to the [Show Fields](#) mode, except that it will show model fields string using the secondary [key fields](#).



Custom Case Information Display Local Menu

By right-clicking on the tabs representing the various sheets of the [custom case information display](#), a local menu appears giving you the following options.

- Rename Sheet – this is only available if you click directly on a specific tab. Choose it to rename the sheet.
- Delete Sheet – this is only available if you click directly on a specific tab. Choose it to delete the sheet.
- New Sheet – Choose this to add a new sheet
- Save All Sheets... – Choose this save all the information on the sheets to an auxiliary file. For the fields one the display, all information will be saved to the auxiliary file using [Primary Key Fields](#) unless the Custom Case Info Mode is set to Show Fields Secondary. In this case, they will be saved using the secondary key fields.
- Custom Case Info Mode – Choose this to open a submenu that allows you to change the mode.
- Tabs Position – Choose this to open a submenu that allows you to change the location of the tabs relative to the sheets
- Plain Text Change Data Color – Choose this to change the color which is used to denote Plain Text cells when the Custom Case Info Mode is set to [Change Field Data](#). The default color is dark gray.

By right-clicking on one of the Custom Case Information Display sheets, you will open a local menu that has many options which are the same as other [Case Information Displays](#). There are a few extra options listed as follows

- Insert Row – Choose this to insert a new row at the location of the presently selected cell
- Delete Row – Choose this to delete rows that are part of the present selection
- Insert Column – Choose this to insert a new column at the location
- Delete Column – Choose this to delete columns that are part of the present selection
- Delete Cell – Choose this to delete the presently selected cells
- Custom Case Info Menu – Choose this to open the same menu available by right clicking on the tabs representing the various sheets

User-Defined Case Information Displays

While it is normally sufficient to customize the case information displays that comes with the program, it is sometimes convenient to create different sets of columns, filtering, sorting, etc... for the same object type. One set of generator fields may be convenient for studying voltage regulation problems, while another set of generator fields may be convenient for looking at MW-related information. This ability is provided using the User-Defined Case Information Displays. This provides the ability to switch between different sets of columns, filters, sorting etc... without requiring repeated customizations.

Note: User-Defined Case Information Displays will not show any of the various Record-Specific Actions which often appear in the [Case Information Toolbar Records Menu](#). To access these special options, the normal case information display must be used instead.

The management of User-Defined Case Information Displays is all done from the [Model Explorer: Explore Pane](#).

Right-Click on Explore Pane to insert a User-Defined Case Information Display

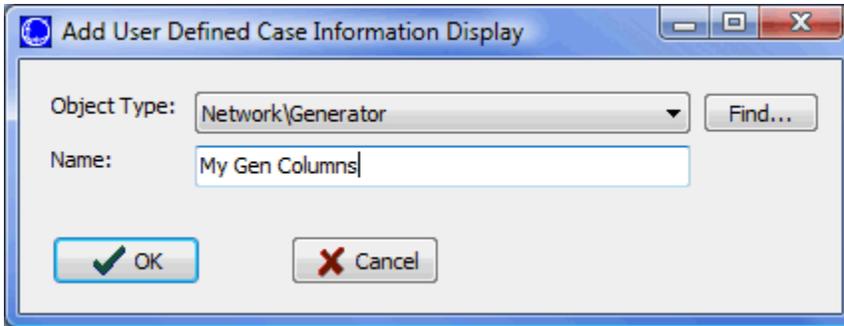
Number of Bus	Name of Bus	ID	Status	Gen MW	Gen Mvar	Set Volt	AGC	AVR	Min MW	Max MW	Min Mvar	Max Mvar	Cost Model	Part. Factor
1	46671 ALDER11	1	Closed	10.00	6.02	1.02899	NO	YES	0.00	26.00	0.00	13.00	None	26.00
2	46672 ALDER12	1	Closed	10.00	6.02	1.02899	NO	YES	0.00	26.00	0.00	13.00	None	26.00
3	47734 SH-P 30	30	Open	0.00	0.00	1.01594	NO	YES	0.00	45.00	-15.00	35.00	None	45.00
4	47735 SH-P 40	40	Open	0.00	0.00	1.01594	NO	YES	0.00	45.00	-15.00	35.00	None	45.00

List of User-Defined Case Information Displays

- Generator: My Gen Colum...
- Load: My Load Colu...

Inserting User-Defined Case Information Displays

To define a User-Defined Case Information Display, right-click on the [Model Explorer: Explore Pane](#), and choose **Insert User-Defined Case Info**. On the dialog which appears click **Find...** to change the **Object Type** and then enter a **Name** for the new display.



Viewing User-Defined Case Information Displays

A list of User-Defined Case Information Displays are shown in the folder **User-Defined** at the bottom of the [Model Explorer: Explore Pane](#).

Removing User-Defined Case Information Displays

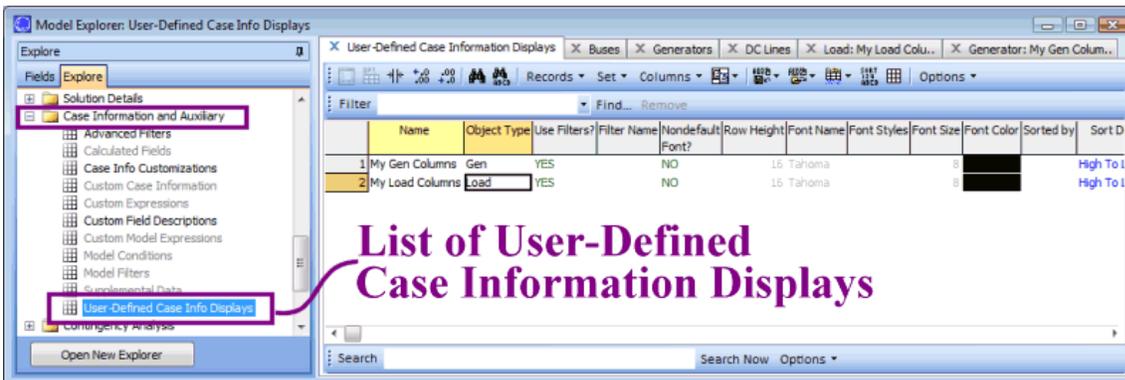
To remove a User-Defined Case Information Display, right-click on its entry under the **User-Defined** folder on the Explore Pane. Then choose **Remove User-Defined Case Info**.

Customizing User-Defined Case Information Displays

The customization of a User-Defined Case Information Display is exactly the same as all other case information displays. See the [Configuring Case Information Displays](#) for more information.

Saving Settings for User-Defined Case Information Displays

Settings for the User-Defined Case Information Displays can be saved in the [Auxiliary File Format](#). To save these, in the Explore Pane, go to the Case Information and Auxiliary/User-Defined Case Info Displays. This will open a case information display showing the user-defined case information displays. Then save to an auxiliary file using the [Case Information Toolbar: Save Auxiliary Files Menu](#).



Case Description

The Case Description Dialog allows you to enter a text description of a case. The portion of the description that is saved with the case varies with case type:

PowerWorld Binary (*.pwb) an unlimited number of lines are allowed

PTI Raw Data format (*.raw) Two lines of text

GE EPC Data format (*.epc) The case title and all the pre-title comments are read in as the case description

PowerWorld Case (*.pwc) No case description supported

IEEE Common Format (*.cf) No case description supported

In PowerWorld Viewer, these descriptions are read-only.

To display the Case Description Dialog to the [Case Information](#) ribbon tab, and choose **Case Description** from the **Case Data** ribbon group.

Case Summary

The Case Summary Display provides a summary of the current case. Note that there are no enterable fields on the display. To display the Case Summary, go to the [Case Information](#) ribbon tab, and choose **Case Summary** from the **Case Data** ribbon group. The fields shown on this display include:

Number of Devices in Case shows the number of each of the following device types:

Buses

Total number of buses in the case. Use the [Bus Display](#) to see a listing of these buses.

Generators

Total number of generators in the case. Use the [Generator Display](#) to see a listing of these generators.

Loads

Total number of loads in the case. Use the [Load Display](#) to see a listing of these loads.

Switched Shunts

Total number of switched shunts in the case. Use the [Switched Shunt Display](#) to see a listing of these switched shunts.

Trans. Lines (AC)

Total number of transmission lines in the case. Use the [Line/Transformer Display](#) to see a listing of these lines.

LTCs (Control Volt)

Total number of Load Tap Changing transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

Phase Shifters

Total number of Phase Shifting transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

Mvar Controlling

Total number of Mvar Controlling transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

Series Capacitors

Total number of Series Capacitors in the case. These devices are treated the same as any other transmission line, and therefore can be found listed as a transmission element in the [Line/Transformer Display](#).

2 Term. DC Lines

Total number of two-terminal dc transmission lines in the case. Use the [DC Transmission Line Display](#) to see a listing of these dc lines.

N-Term. DC Lines

Total number of multi-terminal dc transmission lines in the case. Use the [DC Transmission Line Display](#) to see a listing of these dc lines.

Areas

Total number of areas in case. Use the [Area Display](#) to see a listing of these areas.

Zones

Total number of zones in the case. Use the [Zone Display](#) to see a listing of these zones.

Islands

Total number of islands in the case. An island is a group of buses that are interconnected through ac transmission lines or transformers but are isolated from the rest of the system. Each island must have a slack bus. In Simulator, use the Power Flow Solution tab of the [Simulator Options display](#) to specify whether multiple islands are allowed.

Interfaces

Total number of interfaces in the case. An interface is a grouping of tie line objects between area objects. In Simulator, use the [Interfaces Display](#) to open the [Interface Dialog](#) to define and modify interface objects.

Injection Groups

Total number of injection groups in the case. An injection group is a collection of loads and generators (objects that inject power into a network). In Simulator, use [Injection Groups Display](#) to open the [Injection Groups Dialog](#) to define and modify injection groups.

Case Totals

Summarizes the total load, generation, shunt compensation, and losses for the case. Positive shunt compensation denotes shunt load, whereas negative shunt compensation indicates a shunt injection (such as shunt capacitance). The case totals fields are valid only when the current case is solved.

Generator Spinning Reserves

The total difference between the present total generator output versus the total maximum possible output of all in-service generation.

Slack Buses:

The slack bus or buses are listed showing the bus name, number, and area name and number. One slack bus is required for each island.

Case Pathname

Full file name of the current case.

Power Flow List

The Power Flow List shows detailed information about the system's power flows in a more traditional text-based form. This information is intended for users who would like detailed flow information about the power flow. Including the per unit voltage at the bus, the bus' load and generation, and flows on all lines and transformers emanating from the bus. The content of this display (i.e. which buses are included in the list) is governed by the [area/zone/owner filters](#).

To show this display go to the [Case Information](#) ribbon tab, and choose **Power Flow List** from the **Case Data** ribbon group.

To view flows at just a few select buses you may want to use the [Quick Power Flow List](#) instead. For large systems with no area/zone filtering set, it may take Simulator a long time to generate the complete Power Flow List. Note also that this display can show a maximum of 32,767 lines of text. If this limit is exceeded, Simulator will generate a resource error. Either use the [area/zone/owner filters](#) to limit the number of devices shown on this display, or use the Quick Power Flow List to focus on a few selected buses of interest.

The Power Flow List allows you to navigate through the system's buses rather easily. You can also use the display to show the flows for a bus' neighbor by double-clicking on the line that reads "TO nnnnn...", where nnnnn is the number of the bus you would like to see. The display is then positioned at this bus. If the bus is in an area and/or zone whose area/zone filter is not set, the area/zone filter is set automatically. In this way, you can inspect the system bus by bus.

The display can also be used to quickly display the dialog box associated with different bus objects. Place the cursor on the desired device, depress the CTRL key and then left click. The corresponding bus, load, generator, or shunt information dialogs will be displayed.

The Power Flow List also has its own local menu, which can be viewed by clicking the right mouse button on the display. Select **Change Font** to modify the style and size of the display's font. Select **Refresh** to ensure that the display's contents concur with the current system state. To skip to particular bus in the list, click **Find Bus**, which will open the [Find Bus Dialog](#). To display the information dialog for the currently selected branch, bus, load, or shunt, select **Display Object Dialog**. To print the display, choose the **Print** local menu option. Choosing **Copy** enables you to copy the display into the Windows clipboard, from where the information can be pasted into another application. Finally, select **Close** to close the display.

When printing the display you can either send the results directly to the printer or save them to a text file. To save the results in a text file on the Print Dialog select the **Save to File** option shown in the lower left corner of the dialog.

For each bus, the following items are shown:

Bus

Shows the bus' number, name, and nominal voltage in kV. The next four fields are the MW, MVar, MVA and percentage headers for subsequent rows. The next fields specify the per unit voltage magnitude, voltage angle in degrees, the bus' area number and the bus' area name. Depress the CTRL key and then left click on this line to display the [Bus Dialog](#).

Generator

For each generator at the bus, the Power Flow List shows the generator's ID (immediately after the keyword GENERATOR) and the power output of the generator in MW. Following this is generator's reactive power output in Mvar. A single character is shown immediately after the Mvar field. An 'R' indicates that the generator is regulating the bus voltage, 'H' indicates that the generator is at its high reactive power limit, 'L' indicates that the generator is at its low reactive power limit, and a

blank suggests that the generator is set off of AVR. The last field in the GENERATOR item is the MVA output of the generator. Depress the CTRL key and then left click on this line to display the [Generator Dialog](#). If no generators are connected to the bus, this item will be absent from the display.

Load

Shows the total power consumed by each load at the bus. If no loads are present at the bus, this item will be absent from the display. Depress the CTRL key and then left click on this line to display the [Load Dialog](#).

Shunt

Shows the total power for the fixed shunts at the bus. Positive shunt values denote shunt load, while negative shunt quantities indicate injection. If no shunts are connected to the bus, this item will be absent from the display.

Switched Shunt

Shows the total power for the switched shunts at the bus. Depress the CTRL key and then left click on this line to display the [Switched Shunt Dialog](#). If no switched shunts are located at the bus, this item will be absent from the display.

Lines and Transformers

For each line or transformer coming into the bus, the Power Flow List shows the line's flow and percentage loading. For transformers, the off-nominal tap ratio and phase shift angle in degrees are also shown. Immediately to the right of the off-nominal tap ratio is a two-character designation indicating the tapped side of the transformer: 'TA' indicates that the bus is on the tapped side, while 'NT' identifies the bus as residing on the side without the tap. You can left-click on this field to immediately reposition the bus to the other end of the line or transformer.

Depress the CTRL key and then left click on this line to display the [Transmission Line/Transformer Dialog](#). Simply left-click (without holding the CTRL key) to navigate through the system from bus to bus.

Quick Power Flow List

The Quick Power Flow List provides a convenient means of viewing a listing of the flows at individual buses in the system. The format and control of the Quick Power Flow List is generally the same as that of the [Power Flow List](#), except that the Quick Power Flow List displays results for just the desired bus or [range of buses](#).

You can access this Quick Power Flow List in a number of different ways:

- Go to the [Case Information](#) ribbon tab, and choose **Quick Power Flow List** from the **Case Data** ribbon group
- From most of the Case Information Displays, right-click to invoke the display's local menu, and select *Quick Power Flow List*.
- From the [Bus Information Dialog](#) click on the *View All Flows at Bus* button.
- From the oneline diagram, right-click on the [bus symbol](#) to display the bus' local menu, and select *Quick Power Flow List*.

This display is automatically created if it is not already shown. Information on subsequent buses appears at the bottom of the display.

As with the Power Flow List, you can navigate through the system bus-by-bus by double-clicking on the lines that begin with "TO nnnnn ...," where *nnnnn* is the number of the bus you would like to investigate. Information for that bus will appear at the bottom of the display.

Like the Power Flow List, the Quick Power Flow List has a local menu that is accessed by right-clicking on the display. Among the things you can do from the local menu is to display the bus, branch, generator, load, or shunt corresponding to the currently selected record by selecting **Display Object Dialog**. You can also navigate through the system bus-by-bus just as you can do through double-clicking by choosing **Goto Line Bus** from the local menu.

Outages

The outages display, available by choosing **Solution Details > Outages** in the [Model Explorer](#), presents a tabular listing of devices that are currently out-of-service in the load flow model. The display contains pages that resemble case information displays for [branches](#), [generators](#), [loads](#), [switched shunts](#), [buses](#), and [Multi-section lines](#). All devices listed in the pages on this display are the out-of-service elements.

Area Display

The Area Display houses data about each area in the case. The Area Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated areas. You can also sort the area records by clicking on the heading of the field by which you want to sort.

To show the area records display, select **Aggregations > Areas** from the [Model Explorer](#) or Aggregation > Areas from the Case Information ribbon group under the [Case Information](#) ribbon tab.

By default, the area records display contains the following fields:

Number, Name

Area's number, between 1 and 9999, and its alphanumeric identifier. Simulator supports names of any character length. However, when writing out names to file formats with limitations on name length, the area names will be truncated at their maximum supported length.

AGC Status

Area's automatic generation control status. This field indicates whether or not the area's generation is changing automatically to control the area interchange. See [Area Control](#) for more details. You can toggle the value of the area's AGC Status (except in Viewer) by left clicking on the entry. Valid entries in this field include:

Off AGC Area is off AGC. Generation must be adjusted manually to meet changes in load and losses. If it is not, the system slack will be forced to pick up the balance.

Part AGC Area is on AGC, with generation dispatch controlled by its units' [participation factors](#).

ED Area is on economic dispatch control so that generation is dispatched in order of least cost.

OPF Area is on OPF control (only used with Simulator OPF). This option is only used with Simulator OPF. When the case is solved using the OPF the area controls are changed by the OPF to maintain area power balance. During non-OPF solutions this option is equivalent to Off AGC.

Area Slack Area is on Area Slack control. This option is only available for an area if you have already specified an area slack bus number in the area's [information dialog](#).

Gen MW

Total real power generation in the area in MW.

Load MW

Total real power load in the area in MW.

Tot Sched MW

The net of the base and scheduled transactions between the area and all other areas, with exporting power indicated as a positive value. The interchange for an area is set on the [Area Information Dialog](#); the MW transactions for an area can be viewed using the specified [MW Transactions display](#).

Int MW

The actual interchange between this area and all other areas, with exporting power positive. If an area is on AGC control, its actual interchange should match its scheduled interchange.

ACE MW

The area control error in MW. This is the amount of MW flow difference between the actual MW interchange and the desired MW interchange. A positive value means the super area is generating and exporting excess MW's, and a negative value means the super area is under-generating and importing too many MW's.

Lambda

The area's marginal cost. This marginal cost is relevant only when an Economic Dispatch (not OPF) solution has been run. Theoretically, if losses are ignored, an area operates most economically if all generators operate at the same incremental cost. This common incremental cost is the area's *lambda*, or marginal cost.

The local menu of the area records display has an additional option labeled *All Area Gen IC Curves*. Select this activity to generate a plot showing the incremental cost curves for all units located in a particular area.

Loss (MW)

Total real power losses for the area.

Auto Shunts

Determines whether switched shunts for the area are available for automatic control. You can use this field to disable all the switched shunts in an area. Click on this field to toggle its value. Click on the *Toggle All Yes* or *Toggle All No* local menu options to set the auto shunts property for all switched shunts. Note that a switched shunt is available for automatic control only if it meets three conditions: 1) its control mode property is set to *Automatic* (see [Switched Shunt Information Dialog](#)); 2) its associated area's *Auto Shunts* property is set to *Yes*; and 3) the *Disable Switched Shunt Control* option on the Power Flow Solution tab of the [Simulator Options Dialog](#) must not be checked.

Auto XF

Determines whether tapped transformers for the area are available for automatic control. You can use this field to disable all the transformers in an area. Click on this field to toggle its value. Click on the *Toggle All Yes* or *Toggle All No* local menu options to set all entries in this column. Note that three conditions must be met for a transformer to be used for automatic control: 1) its *Auto* field must be set to *Yes* (see [Transformer Modeling](#) for details); 2) its associated area's *Auto XF* property is set to *Yes*; and 3) the *Disable Transformer Control* option on the Power Flow Solution tab of the [Simulator Options dialog](#) must not be checked.

Unspec. MW Inter.

Shows the total amount of interchange for the area that is listed as Unspecified. This means that some of the interchange is known in magnitude and direction (import or export) but not which other area(s) is involved in the interchange. The sum of all area's unspecified interchange in the case must equal 0 for a balanced system.

Zone Display

The Zone Display provides information about all the zones in the case. Similar to the [Area Display](#), the Zone Display provides a means of dividing up a power system. System results can then be summarized by zones using this display. Buses can be assigned to zones independent of their area assignments. Thus a single area could contain multiple zones, or a single zone could span multiple areas. The zone number for each bus is shown on the [Bus Dialog](#). In the Edit Mode, groups of buses can be easily moved from one zone to another using the [Zone Dialog](#).

The Zone Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated zones. You can call up the [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about a representative bus in the zone. You can also sort the zone records by clicking on the heading of the field by which you want to sort.

To show this display, click on **Aggregations > Zones** in the [model explorer](#).

The display contains the following fields by default:

Zone Number, Zone Name

Zone's number, between 1 and 9999, and its alphanumeric identifier. Simulator supports names of any character length. However, when writing out names to file formats with limitations on name length, the zone names will be truncated at their maximum supported length.

Load MW, Load MVR

Total real and reactive power load in the zone.

Gen (MW), Gen (Mvar)

Total real and reactive power generation in the zone.

Loss MW, Loss MVR

Total real and reactive power loss in the zone. Losses are computed by summing the losses of the individual transmission lines and transformers in the zone. Because of shunt charging, these devices can also generate reactive power. Therefore, reactive power losses may actually be negative.

Int MW, Int Mvar

Net interchange of real and reactive power with all other zones. Exported power is assumed to be positive.

Load Mult MW

Shows the current load MW multiplier for the zone. All load in the area is scaled by this multiplier.

Tie Lines between Areas Display

The Tie Lines between Areas Display identifies all area tie lines in the case. Tie lines are not a separate data record that can be created in Simulator, but instead provide a look at all the devices that connect two areas together in the power system model. Tie lines are important because the [Area Control](#) algorithms used by Simulator often enforce a constraint which requires that the sum of the flow on the tie lines must equal the net export scheduled by the area (This is often called the ACE equation).

There are several types of devices that can represent tie lines. These include the following:

- Transmission branch (Branch) : a transmission line that connects two buses which belong to two different areas is a tie line.
- Two-terminal DC transmission lines (DCLine) : a DC transmission line that connects two buses which belong to two different areas is a tie line.
- Multi-terminal DC transmission line (MTDCRecord) : For every multi-terminal DC transmission line which has converters connected to more than one area is a tie line.
- Load (Load): A load can be assigned to a different area than its terminal bus. This is not typical but is sometimes done to model loads at lower voltage level where more than one entity has load that connects to the same power system bus. In such a situation this load represents a tie line.
- Generator (Gen): Similarly to the load, a generator can be assigned to a different area than its terminal bus. Again this is considered a tie line.
- Switched Shunt (Shunt): Similarly to the load, a switched shunt can be assigned to a different area than its terminal bus. Again this is considered a tie line.

Note: The easiest way to see which loads, generators, or switched shunts are assigned in this way is to use this Tie Lines between Areas Display. You can also however add separate columns using the display/column options on the Load, Generator or Switched Shunt Case Information Display. For instance for loads there area columns for Area Num of *Bus* and Area Num of *Load*. Normally these are equal, but they will not be for tie line loads.

Note: Be careful when working with cases opened as RAW files in which there are loads that represent tielines. See Add and Remove of transactions due to tie-line loads in [Simulator Options: File Management](#) for more information.

The Tie Lines between Areas Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated device type. You can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the tie line information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Tie Lines Display according to the constraints of the [area/zone/owner filters](#).

To call up the Tie Lines between Areas Display, select **Aggregation > Tie Lines between Areas** from the [Model Explorer](#).

The Tie Lines between Areas Display contains the following fields by default:

Tie Type

Indicates type of device which the tie line represents: Branch, Load, Gen, Shunt, MTDCRecord, or DCLine.

Area Name, Number, Name

The Bus Number, Name and Area Name of the bus to which the device is attached.

Far Area Name, Far Number, Far Name

The Bus Number, Name and Area Name of the other end of the device.

Ckt

The circuit ID for transmission lines or DC Lines, the device ID for generators, loads, and switched shunts.

Meter MW and Mvar

The MW and Mvar flow from the point represented by Number/Name towards the far Number Name. For transmission lines and DC lines the metered point on the line is determined by the Metered End which is stored with the device. For instance on a transmission line, there is a check box for From End Metered which is available on the [Transmission Line/Transformer Option \(Edit Mode\)](#) dialog. Changing this setting with the device will affect the Meter values slightly.

Status

This is the Open/Closed status of the tie line device.

Lim MVA

For tie lines that represent transmission lines this is the limit of the device.

MW and Mvar Loss

For tie lines that represent transmission lines this is the MW and Mvar loss across the device.

Tie Lines between Zones Display

The Tie Lines between Zones Display is essentially identical to the [Tie Lines between Areas Display](#). The only difference is that it displays devices that connect two zones together instead of areas. See the [Tie Lines between Areas Display](#) for more details.

Super Area Display

The Super Area Display identifies any super areas that have been defined for the case. Super areas are groups of areas whose generators are dispatched as a coordinated group. Super areas can thus be useful for modeling the operation of independent system operators or power pools.

Super areas cannot be inserted into a case from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Instead, a super area can be defined when modifying or creating an area simply by typing the name of a new super area in the *Super Area* dropdown box on the [Area Information Dialog](#).

The Super Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated super areas. You can call up the [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about representative bus in the super area. You can also sort the super area records by clicking on the heading of the field by which you want to sort.

To show this display select **Aggregation > Super Areas** in the [Model Explorer](#).

The display contains the following fields by default:

Super Area

The name of the super area.

AGC Status

The Super Area may operate without automatic generation control (AGC Status = *Off AGC*), with participation factor control (AGC Status = *Part. AGC*), or according to an economic dispatch (AGC Status = *ED*). This is a toggleable field.

Use Area PF

Indicates whether to use the areas participation factors when the super area is to operate with participation factor control (Part. AGC).

Num Areas

Indicates the number of areas defined as being part of the super area. Areas are added to super areas using the Super Area dropdown box on the [Area Information Dialog](#).

Gen MW

Total MW injection from all the generators in the super area.

Load MW

Total MW load demanded in the super area.

Tot Sched MW

Total scheduled MW interchange with other areas or super areas.

ACE MW

The area control error in MW. This is the amount of MW flow difference between the actual MW interchange and the desired MW interchange. A positive value means the super area is generating and exporting excess MW's, and a negative value means the super area is under-generating and importing too many MW's.

Lambda

The super area's marginal cost.

Loss MW

Indicates the real power losses incurred within the super area.

ED Use PF

Indicates whether the power flow engine will calculate loss penalty factors in computing the economic dispatch solution for the super area. If loss penalty factors are not calculated, then the economic dispatch is calculated assuming that the super area is lossless. Otherwise, the economic dispatch solution incorporates losses. The penalty factors gauge the sensitivity of the area's losses to changing injection at specific generators. The option to calculate loss penalty factors is relevant only when the super area operates according to *Economic Dispatch Control*. Usually, if the system's cost curves are relatively flat, the inclusion of losses in the solution will not have much of an effect on the dispatch.

Bus Display

The Bus Display presents data describing each bus in the case. The Bus Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated bus. You can also sort the bus records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Bus Records Display according to the constraints of the [area/zone/owner filters](#). Finally, when Simulator is in [Edit Mode](#), the case information toolbar allows you to add new buses to or remove existing buses from the system.

To show the bus records display, select **Network > Buses** from the [Model Explorer](#) or from the Case Information ribbon group under the [Case Information](#) ribbon tab.

By default, the bus records display contains the following fields:

Number and Name

Bus number between 1 and 2,147,483,647 (equals 2^{31} minus 1), and its alphanumeric identifier.

Area Name

Alphanumeric identifier of the bus' area.

Nom kV

The nominal base voltage of the bus in kV.

PU Volt

Bus' per unit voltage magnitude.

Volt (kV)

Bus' actual voltage magnitude in kV. This is the per unit voltage magnitude multiplied by the bus' nominal voltage.

Angle (Deg)

Bus' voltage angle in degrees.

Load MW, Load Mvar

Total real and reactive load at the bus. If no loads are located at the bus, these fields are blank.

Gen MW, Gen Mvar

Total real and reactive generation at the bus. If no generators are located at the bus, these fields are blank.

Switched Shunt Mvar

Total switched shunt device reactive power injection at the bus.

Act G Shunt MW, Act B Shunt Mvar

Total real and reactive fixed bus shunt injections.

Area Num

The area number in which the bus is located.

Zone Num

The zone number in which the bus is located.

Remotely Regulated Bus Display

The Remotely Regulated Bus Display provides information about all buses that are remotely regulated by one or more generators, transformers, or switched shunts. The bus that a generator regulates is specified in the Edit Mode using the [Generator Dialog](#). Whenever a generator is regulating a bus that is not its terminal, it is considered to be remotely regulating that bus. The bus that is remotely regulated, along with the regulating generators, will appear on this display. The same logic is true for transformers and switched shunts which may regulate remote buses also.

The Remotely Regulated Bus Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view [the information dialog](#) of its associated buses. You can find a specific remotely regulated bus, and you can learn more information about a particular remotely regulated bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the remotely regulated bus information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To call up the Remotely Regulated Bus Display, open the [Model Explorer](#) and select **Remotely Regulated Buses** under the Solution Details folder.

This Remotely Regulated Bus Display contains the following fields by default:

Number, Name, Area Name

Number, name, and area name for the bus that is being remotely regulated.

PU Volt

Per unit voltage magnitude for the bus.

Set Volt

Setpoint voltage for the bus. When a bus is being remotely regulated by a set of generators, the generators vary their reactive power output to maintain the voltage at the bus at the setpoint value. You can enter a new value for this field. Changing the setpoint voltage here changes the setpoint voltage for all the generators that are remotely regulating this bus.

Volt Diff

Per unit difference between the actual voltage magnitude and the set point voltage magnitude.

AVR

Combined automatic voltage regulation (AVR) status for all the generators remotely regulating this bus. If AVR is *No*, no generators regulate voltage; if AVR is *Yes*, all the available generators are regulating; if AVR is *Mixed*, some generators regulate voltage and some do not. Regulation of individual generators can be specified using the [Generator Display](#). You can toggle this field between "Yes" and "No" by clicking on it.

Total Mvar

Total of the reactive power being supplied by all the generators remotely regulating the bus.

MVR Min, MVR Max

Total of the minimum and maximum reactive power limits for all the generators remotely regulating the bus.

Remote Regs (gen)

Provides a comma-separated list of all the generators which regulate this bus.

Rem Regs (XFMR)

Provides a comma-separated list of all the transformers which regulate this bus.

Rem Regs (SS)

Provides a comma-separated list of all the switched shunts which regulate this bus.

Bus Mismatches Display

The Bus Mismatches Display lists the real and reactive mismatches at each bus. The bus mismatches are defined as the difference between the power entering the bus and the power leaving the bus. A power flow case is considered solved when all the bus mismatches are below the convergence tolerance specified on the Power Flow Solution page of the [Simulator Options](#) dialog.

Most of the time you will not need to be concerned about the bus mismatches. If the power flow solves, the mismatches are guaranteed to be below the desired tolerance. However, advanced users will find this display useful in determining the cause when a power flow diverges.

The Bus Mismatch Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated bus. You can find a specific bus mismatch using the name or number of the bus, and you can learn more about the bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the bus mismatch information by clicking on the heading of the field by which you want to sort.

To show this display select **Solution Details > Mismatches** from the [Model Explorer](#). You can view bus mismatches only when the application is in Run Mode.

Also note that there is a special feature in the Records menu of the case information toolbar for the mismatch display. This option is **Zero-Out Mismatches**. Choosing this options will cause Simulator to insert fictitious B and G Bus Shunt values to force the mismatch at every bus to zero.

The Bus Mismatch Display contains the following fields by default:

Number, Name, Area Name

The number and name of the bus and the name of the area in which it is located.

MW Mismatch, MVR Mismatch, MVA Mismatch

The real and reactive mismatches at each bus, and the total complex power mismatch.

Substation Records Display

The Substation Records Display presents data describing each substation in the case. The Substation Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the information dialog of its associated substation. You can also sort the substation records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Substation Records Display according to the constraints of the [area/zone/owner filters](#). Finally, when Simulator is in [Edit Mode](#), the local menu allows you to add new substations to or remove existing substations from the system. Note: a substation is considered in an area/zone if any single bus in the substation is in the area/zone. To show the substation records display, select **Aggregation > Substations** in the [Model Explorer](#).

By default, the substation records display contains the following fields:

Sub Num

An integer identifier for the substation.

Sub Name, Sub ID

Two alphanumeric identifiers for the substation.

Area Name, Zone Name

The names of the area and zone of the buses in the substation. If some of the buses in the substation are in different areas or zones, then this is the most common area or zone.

of Buses

The number of buses inside the substation.

Nom kV

Nominal kV of the highest voltage bus(es) in the substation.

Gen MW, Gen MVR

Total real and reactive generation at the substation. If no generators are located at the substation, these fields are left blank.

Load MW, Load MVR

Total real and reactive load at the substation. If no loads are located at the substation, these fields are left blank.

Shunt MW, Shunt MVR

Total real and reactive shunt values at the substation. If no shunts are located at the substation, these fields are left blank.

Generator Display

The Generator Display presents data describing each generator in the case. The Generator Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Records Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

Some columns which can be added to the generator records are related to the post-contingency solution options. For more information about these columns see the sections "Define Generator Line Drop and Reactive Current Compensation" and "Define Generator Maximum MW Responses in Post-Contingency" in the [Advanced Modeling Options](#) for [Contingency Analysis](#).

To show the generator records display, select **Network > Generators** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of the bus to which the generator is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for the bus.

ID

Single character ID used to distinguish multiple generators at the same bus. This default value for this field is '1'.

Status

Displays the Open / Closed status of the generator. This field is a toggleable field.

Gen MW, Gen Mvar

The real and reactive power output of the generator. If the generator is on AVR control, the reactive power is set automatically.

Set Volt

Per unit setpoint voltage for the generator. When a generator is on AVR control, the reactive power output of the generator is varied automatically in order to maintain the regulated bus voltage at this value. The regulated bus is usually, but not always, the generator's terminal bus. Use the [Generator Dialog](#) to see the regulated bus number.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

AVR

Designates whether the generator will vary its reactive power output to maintain a constant terminal voltage. If the *AVR* property is set to *Yes*, the generator is on automatic voltage regulation (AVR)

control. When a generator is on AVR control, its reactive power output is varied automatically to keep the regulated bus voltage at the **Set Volt** value. AVR is limited by the generator's reactive power limits. You can click on this field to toggle its value (except in Viewer).

Min MW, Max MW

Minimum and maximum allowable real power output of the generator.

Min Mvar, Max Mvar

Minimum and maximum allowable reactive power output of the generator.

Cost Model

The type of cost model the generator is currently set to use. Cost models are necessary for performing economic analysis, such as Economic Dispatch or [Optimal Power Flow](#).

Part. Factor

Generator's participation factor. Participation factors are used to determine how AGC generators participate in area control when their area is on participation factor control. Please see [Area Control](#) for more details.

Generator/Load Cost Models

The Generator/Load Cost Models option from the Case Information menu allows you to choose to view detailed information on generator cost curves or load benefit curves. This cost information can be very important when solving an [Optimal Power Flow](#) or [Security Constrained Optimal Power Flow](#) solution.

The Generator/Load Cost Models option has four submenu options to choose from:

[Generator Cubic Cost Models](#)

[Generator Piecewise Linear Cost Models](#)

[All Generator Cost Models](#)

[All Load Benefit Models](#)

Generator Cost Models Display

The Generator Cost Models Display presents detailed cost information for each generator in the case, regardless of what type of cost curve has been entered for the generator. The Generator Cost Models Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Cost Models Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator piecewise linear and cubic cost model display, select **Network > Generators > Cost Curves All** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the generator is attached.

Area Name of Gen

Name of the area to which the generator belongs. The generator can be assigned to an area different than the area to which its terminal bus is assigned.

ID

Single character ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the generator. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power output of the generator.

Min MW

Minimum MW output of the generator.

Max MW

Maximum MW output of the generator.

Cost Model

The type of model this generator is currently using. Can be Cubic, Piecewise Linear or None.

IOA, IOB, IOC, IOD

Parameters used to model the cost characteristic of the generator. Please see [Generator Cost Information](#) for details. Please note that these values can be saved/loaded using the [Generator Cost Data](#) auxiliary file.

These fields will be disabled unless the Cost Model type is set to Cubic.

Fuel Cost

The fuel cost of the type of fuel for the generator.

Variable O&M

Operations and Maintenance costs for the generator.

This field will be disabled unless the Cost Model type is set to Cubic.

Fuel Type

An informational field that can be set to the type of fuel the generator uses.

Unit Type

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Fixed Cost

The fixed operating cost of the generator.

MWh Break x, MWh Price x

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW output value of the generator. The MWh Price value is the corresponding marginal cost of producing an additional MW of power at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal price of the unit from that break point location to the maximum output of the generator.

A requirement of the piecewise linear cost curve is that it must be convex, meaning the next MWh Price must be higher than the previous MWh Price.

These fields will be disabled unless the Cost Model type is set to Piecewise Linear.

Generator Cubic Cost Display

The Generator Cubic Cost Display presents detailed cost information for each generator in the case set to use a cubic cost model. The Generator Cubic Cost Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Cubic Cost Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator. To show the generator cubic cost display, select **Network > Generators > Cost Curves Cubic** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the generator is attached.

Area Name of Gen

Name of the area to which the generator belongs. The generator can belong to an area which is different than the area of which its terminal bus is a member.

ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the generator. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power output of the generator.

IOA, IOB, IOC, IOD

Parameters used to model the cost characteristic of the generator. Please see [Generator Cost Information](#) for details. Please note that these values can be saved/loaded using the [Generator Cost Data](#) auxiliary file.

Fuel Cost

The fuel cost of the type of fuel for the generator.

Variable O&M

Operations and Maintenance costs for the generator.

Fuel Type

An informational field that can be set to the type of fuel the generator uses.

Unit Type

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Cost \$/Hr

Operating cost for the generator in \$/hr.

IC

Incremental cost to produce an additional MWh. This can be expressed as $dC(P_{gi})/dP_{gi}$, where C denotes the generator's cost of operation in \$/hr and P_{gi} expresses the current MW output of the unit. In a lossless system, the incremental dispatch is equal to the generator's lambda value.

LossSens

Area loss sensitivity - This field is **only calculated when the generator's area is on economic dispatch control**. This field specifies the incremental change in area losses if this generator were to produce one more MW, **with the excess generation absorbed by the system slack**. This may be expressed as $\partial P_{loss}/\partial P_{gi}$. The loss sensitivity is used in calculating the generator's lambda value for the economic dispatch activity.

Generator MW Marg. Cost

The marginal cost of the generator supplying an additional MW of power to the system.

Generator Piecewise Linear Cost Display

The Generator Piecewise Linear Cost Display presents detailed cost information for each generator in the case set to use a piecewise linear cost model. The Generator Piecewise Linear Cost Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Piecewise Linear Cost Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator piecewise linear cost display, select **Network > Generators > Cost Curves Linear** in the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the generator is attached.

Area Name of Gen

Name of the area to which the generator belongs. The generator can belong to an area which is different than the area of which its terminal bus is a member.

ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the generator. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power output of the generator.

Min MW

Minimum MW output of the generator.

Max MW

Maximum MW output of the generator.

Fuel Cost

The fuel cost of the type of fuel for the generator.

Fuel Type

An informational field that can be set to the type of fuel the generator uses.

Unit Type

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Fixed Cost

The fixed operating cost of the generator.

MWh Break x, MWh Price x

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW output value of the generator. The MWh Price value is the corresponding marginal cost of producing an additional MW of power at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal price of the unit from that break point location to the maximum output of the generator.

Load Display

The Load Display presents data describing each load in the case. The Load Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated loads. The toolbar also affords the opportunity to insert new loads into the model or to delete existing ones. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for the load's terminal bus. You can also sort the load information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To show the load display, select **Network > Loads** in the [Model Explorer](#).

The load display contains the following fields by default:

Number, Name

Number and name of bus to which the load is attached.

ID

Two-character ID used to distinguish multiple loads at the same bus; '1' by default.

Status

Either Closed if the load is connect to its bus, or Open if it is not. You can click on this field to toggle its value. If the load is open, the entire load record is dimmed.

MW, Mvar, MVA

Total real, reactive, and complex power for the load. Loads may be both voltage dependent. The total load is the sum of the constant power, constant current, and constant impedance components. See [Load Information](#) and see [Load Modeling](#) for more information.

S MW, S MVAR, I MW, I MVR, Z MW, Z MVR

These six fields describe the composition of the load at the bus assuming 1 pu bus voltage. The SMW and SMVAR fields indicate the constant power portion of the load, the component that does not vary with bus voltage magnitude. The IMW and IMVR fields express the constant current part of the load, which varies in proportion to the bus voltage magnitude. Finally, ZMW and ZMVR indicate the constant impedance portion of the load, which varies with the square of the voltage. The sum of the SMW, IMW, and ZMW fields yields the base MW load at the bus (assuming 1pu voltage), and the sum of the SMVR, IMVR, and ZMVR fields provides the base MVR load at the bus (assuming 1 pu voltage). Please see [Load Modeling](#) for more details on how bus load is modeled.

Load Benefit Models Display

The Load Benefit Models Display presents detailed cost information for each load in the case set to use a piecewise linear benefit model. The Load Benefit Models Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated loads. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the load's terminal bus. You can also sort the load records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Load Benefit Models Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new load into the case or to delete an existing load.

To show the generator piecewise linear cost display, select **Network > Loads > Benefit Curves Linear** in the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the load is attached.

Area Name of Gen

Name of the area to which the load belongs. The load can belong to an area which is different than the area of which its terminal bus is a member.

ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the load. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control, since effectively dispatching a load can be viewed as dispatching negative generation. If the AGC field is set to *Yes*, the load is on automatic generation control (AGC). When a load is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power demand of the load.

Min MW

Minimum MW demand of the load.

Max MW

Maximum MW demand of the load.

Fixed Benefit

The fixed benefit of the load.

Benefit Model

The type of model this load is currently using. Can be either Piecewise Linear or None.

MWh Break x, MWh Price x

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW demand value of the load. The MWh Price value is the corresponding marginal benefit of extracting an additional MW of load at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal benefit of the load from that break point location to the maximum demand of the load.

A requirement of the piecewise linear benefit curve is that it must be concave, meaning the next MWh Price must be lower than the previous MWh Price. In other words, as more load is supplied, the less the benefit it is providing. This is how a load can be dispatched along with generation, according to marginal costs (and marginal benefits.)

These fields will be disabled unless the Cost Model type is set to Piecewise Linear.

Line and Transformer Display

The Line/Transformer Display presents data describing each transmission line and transformer in the case. The Line/Transformer Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated branches. The local menu also affords the opportunity to insert new lines or transformers into the model or to delete existing ones. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for each branch's terminal buses. You can also sort the line and transformer information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Line/Transformer Display according to the constraints of the [area/zone/owner filters](#).

To show the line/transformer display, select **Network > Branches Input** or **Network > Branches State** from the [Model Explorer](#).

The contents of the display depends upon the application's operating mode.

The line/transformer display shows the following fields by default:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side. Right-clicking on either of these fields brings up the display's local menu from which you may select [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about the *from bus*.

To Bus Number and Name

To Bus number and name. Right-clicking on either of these fields brings up the display's local menu from which you may select [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about the *to bus*.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Status

The service status of the branch. This field is toggleable.

Xfrmr

Yes or no field signifying if the branch is a transformer or transmission line. This field cannot be changed.

MW From, Mvar From, MVA From (State display default)

Real, reactive, and complex power flowing into the line at the *from bus*.

In versions of Simulator prior to version 14, these values were always calculated based on the present voltage at a bus and other branch parameters each time they were displayed. This means that a user could change the bus voltage, or impedance, and see a corresponding change in the line flow values. These values are no longer calculated each time that they are displayed. They are updated following a power flow solution, loading in a case, following an update to the system due to OPF control changes, and after creating an equivalent. The exception to this is that when loading in a PowerWorld Binary (*.pwb) file and the flows are stored in the file, the stored flows will be retained instead of updating them based on the present system state. The MW and Mvar values can be set to user-specified values using an auxiliary file or pasting from Excel. This will allow the contouring of flows based on some external source.

Four additional fields have been added that will update each time they are displayed. This means that they will be updated following a voltage change for a bus or other relevant parameter (i.e. impedance) change for a branch. These fields take on the functionality of the flow fields prior to version 14 and are calculated each time that they are displayed. These fields are MW From (Calculated), MW To (Calculated), Mvar From (Calculated), and Mvar To (Calculated). These fields cannot be changed by the user.

Lim MVA, % of MVA Limit (Max) (State display default)

The current MVA limit of the branch, and the amount of the actual flow as a percentage of the MVA limit.

MW Loss, Mvar Loss (State display default)

Real and reactive power losses on the transmission line or transformer. Since reactive power losses include the effect of line charging, the reactive power losses may be negative.

R, X, B (Input display default)

Indicates the branch's resistance, reactance, line charging susceptance in per unit on the system base.

Lim A MVA, Lim B MVA, Lim C MVA (Input display default)

Identifies the first three limit settings. Five additional limits can be set, for which the columns can be added using the Display/Column options from the local popup menu. All limits are expressed in MVA.

See [Transformer Display](#) for viewing transformer specific fields.

Multi-Section Lines Display

To show this display select **Aggregations > Multi-Section Lines** in the [Model Explorer](#).

The Multi-Section Line Display lists the multi-section lines that exist in the case. The Multi-Section Line Display is used to functionally group a number of transmission lines together. They are usually used to model very long transmission lines that require the use of multiple individual transmission line information to be modeled accurately. Simulator then treats each multi-section line as a single device with regard to line status. That is, changing the status of one line in the record changes the status of the other lines in the record as well.

Each multi-section line record consists of the "from" bus, one or more "dummy" buses and the "to" bus. Each dummy bus must have only two lines connected to it, each of which are members of the multi-section line record. See [Multi-Section Line Information](#) for details.

The Multi-Section Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its multi-section line information. The local menu also affords the opportunity to insert new multi-section lines into the model or to delete existing ones when the application operates in the Edit Mode. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for the line's terminal buses. You can also sort the multi-section line information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Multi-Section Line Display according to the constraints of the [area/zone/owner filters](#).

While in Edit mode, the local menu allows two options that are specific to multi-section lines. The options are found under the Records menu item.

Merge Sections

Select this option to eliminate the selected multi-section line record and designate the section lines as single line records.

Renumber Dummy Buses

This option allows renumbering/renaming the dummy buses that are part of a multi-section line by loading a file that has the new designations. The file containing the renumbering should be a text file in the following format:

```
frombus, tobus, circuit // this identifies the multi-section line
dummybusnumber1, dummybusname1
dummybusnumber2, dummybusname2
frombus, tobus, circuit // this identifies another multi-section line
dummybusnumber1, dummybusname1
dummybusnumber2, dummybusname2
dummybusnumber3, dummybusname3
```

The file can either be space or comma delimited. A double slash can be used for a comment. A line with three entries is assumed to be a multi-section line and any two entry lines following that are assumed to be the dummy bus identifiers belonging to that multi-section line. Dummy buses are read for a multi-section line until another three entry line is found, meaning that a new multi-section line has been specified. Dummy buses are identified starting at the from bus of the multi-section line going to the to bus and that is the way that they should be entered in the file. If a multi-section line matching the given identifiers cannot be found, all dummy buses after that multi-section line record are ignored until a new multi-section line specification is found. If an incorrect number of dummy buses is entered for a multi-section line, none of the dummy buses

will be updated. If the a dummy bus number is specified that matches an existing bus that is not another dummy bus, no changes will be made for that dummy bus. If a dummy bus number is specified that matches an existing bus that is another dummy bus, the other dummy bus will be assigned to a new bus number and the current dummy bus will be assigned to the number specified in the file.

The display contains the following fields by default:

NumberFrom, NameFrom

Number and name of the multi-section lines *from bus*. Right clicking on one of these fields invokes the display's local menu from which you can select either [Quick Power Flow List](#) or [Bus View Display](#) to find more information about the *from bus*.

NumberTo, NameTo

Number and name of the multi-section lines *to bus*. Right clicking on one of these fields invokes the display's local menu from which you can select either [Quick Power Flow List](#) or [Bus View Display](#) to find more information about the *to bus*.

Circuit

Two-character circuit identifier for the multi-section line. The first character in the identifier should always be an "&."

Sections

Number of individual lines within the multi-section line record.

Allow Mixed Status

This value defaults to NO and means that all branches in the multi-section line must have the same status. Setting this value to YES means essentially means that the multi-section line is disabled so that opening/closing a branch in the multi-section line will no longer automatically open or close the other lines in the record.

Status

Current status of the record. Note, the status is *Closed* only if **all** the lines in the record are closed, and is *Open* only if **all** the lines in the record are open. Otherwise the status is *Mixed*.

Transformer Display

The Transformer Display identifies all transformers in the case. The data presented in the Transformer Display supplements the data presented in the [Line/Transformer Display](#) by presenting transformer-specific information. Consult the Line/Transformer Display for the transformer flows.

The Transformer Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated transformer. You can find a specific transformer using the names or numbers of its terminal buses, and you can learn more about a particular transformer's terminal buses by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can delete an existing transformer from the case. You can also sort the transformer information by clicking on the heading of the field by which you want to sort.

To show this display select **Network > Transformers Controls** in the [Model Explorer](#).

The Transformer Display contains the following fields by default:

From Bus Number and Name

From Bus number and name. The *From Bus* is the tapped side. You may view either the [Quick Power Flow List](#) or the [bus view display](#) for the *From Bus* from the local menu.

To Bus Number and Name

To Bus number and name. You may view either the [Quick Power Flow List](#) or the [bus view display](#) for the *To Bus* from the local menu.

Circuit

Two-character identifier used to distinguish between multiple transformers joining the same two buses.

Status

The service status of the transformer. This field is toggleable.

Type

Type of transformer. Possible values include

Fixed The tap positions are fixed

LTC The tap ratio changes to regulate bus voltage

Mvar The tap ratio changes to regulate reactive power flow

Phase The phase angle changes to regulate real power flow

Tap/Phase

Indicates the tap ratio for LTC and fixed transformers and the phase shift angle in degrees for phase-shifting transformers.

XF Auto

If the value of this field is *Yes*, the transformer will automatically change its tap or phase angle to keep the regulated value within the specified regulation range, provided that the *Auto XF* field of its associated area is set to *Yes* and transformer tap/phase control has not been disabled for the entire case. The *Auto XF* field of individual areas is set from the [Area Display](#), and case-wide transformer control can be set from the Power Flow Solution tab of the [Simulator Options](#) Dialog. Click on this field to toggle its values.

Reg Bus

For an LTC transformer, this is the number of the bus whose voltage is controlled by the transformer. For phase shifting transformers, the real power is always controlled at the tapped bus.

Reg Value

For an LTC transformer, this is the present per unit voltage at the regulation bus. For a phase shifting transformer, this is the present real power flow through the transformer measured on the "from" (tapped) side.

Reg Error

The error is the difference between the regulated value and the respective limit of the regulation range specified by Reg Min and Reg Max. If the regulated value is within the regulation range, then the error is zero. The error is negative if the regulated value falls below the regulation range, and it is positive if the regulated value exceeds the regulation range.

Reg Min, Reg Max

Minimum and maximum values for the regulation range. For LTC transformers, these fields represent per unit voltage at the regulated bus. For phase shifting transformers, these fields represent actual MW flow through the transformer measured on the "from" (tapped) side. Because transformers use discrete control, the maximum regulation value must be somewhat greater than the minimum value.

Tap Min, Tap Max

For LTC transformers, these fields specify the minimum and maximum tap ranges for the transformer. For phase shifting transformers, these fields specify the minimum and maximum phase shift angle in degrees.

Step Size

The per unit step size for tap changing transformers. This step size is usually determined by dividing the total range of transformer operation by the number of tap positions available.

Three Winding Transformer Display

The Three Winding Transformer Display shows information about all the three winding transformer devices in the case. Three winding transformers are modeled in Simulator as a grouping of two winding transformers, connected at a common midpoint or "Star" bus. The Three Winding Transformer Display is a way to view all the terminal connection points for the three winding transformers and the power delivered at each of the terminals.

The Three Winding Transformer Display is a class of [Case Information Display](#), and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated three winding transformers. When in Edit Mode, you can define new three winding transformers using the **Insert** option, or delete existing three winding transformers using **Delete**. You can also sort the three winding transformer information by clicking on the heading of the field by which you want to sort.

To show this display, open the [model explorer](#) and click on **Network > Three Winding Transformers**.

This display contains the following fields by default:

Pri Bus Num, Sec Bus Num, Ter Bus Num

These are the primary winding, secondary winding and tertiary winding terminal bus numbers for the three winding transformer connections.

Circuit

The circuit identifier for the three winding transformer.

Pri, Sec and Ter MW and MVAr

These fields display the real and reactive power delivered at each of the three winding transformer terminals.

Star Bus Number and Name

These two fields provide the number and name of the internal node star bus used in the three winding transformer equivalent model.

Line Shunts Display

The Line Shunts Display identifies all line shunt devices in the case. Line shunts are similar to [switched shunts](#) in that they are used in power systems either to inject additional MVR into the system (capacitive shunts) or to absorb excess reactive power (inductive shunts). However instead of being connected to a bus directly, they are instead shunt connected to one of the terminals of a transmission line. As a result they can only be in-service if the actual transmission line is in-service. Also, Simulator also does not allow the automatic regulation of a bus voltage using Line Shunts as is possible using a switched shunt. Simulator does however allow line shunts to be individually taken in and out of service. This must be done manually however using the status field of the line shunt.

The Line Shunts Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated line shunts. You can find a specific line shunt using its bus name or number, and you can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can insert a new line shunt into the case or delete an existing shunt. You can also sort the line shunt information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Line Shunts Display according to the constraints of the [area/zone/owner filters](#).

To call up the Line Shunts Display, click **Network > Line Shunts** in the [Model Explorer](#).

The Line Shunts Display contains the following fields by default:

From Number and To Number

Numbers of from and to bus of the transmission branch to which the line shunt is attached. Use the right menu options to inspect either the [Quick Power Flow List](#) or the [Bus View Display](#) for the terminal buses.

Circuit

Circuit identifier for the transmission branch to which the line shunt is attached.

Bus Number Located At

Either the From or To Bus number. Specified the end of the transmission branch to which the line shunt is attached.

ID

A circuit identifier for the actual line shunt. This is needed because there can be multiple line shunts at the same end of the same transmission line.

G, B

Indicates the line shunts nominal conductance and susceptance in per unit on the system base.

Status

Line shunts can be individually taken in and out of service. Do this by changing the status field to Open or Closed.

DC Lines Display

The DC Line Display presents data describing each dc line in the case. The DC Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated dc lines. You can also sort the dc line records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the DC Line Records Display according to the constraints of the [area/zone/owner filters](#).

To show the dc line records display, select **Network > DC Transmission Lines** in the [Model Explorer](#), and click on the DC Lines tab.

By default, the dc line records display contains the following fields:

Number

Each dc line must be assigned a unique number, typically between 1 and 40.

Rect AC Number, Rect AC Name, Rect MW, Rect Mvar

Number and name of the rectifier bus, and the real and reactive power flow from the rectifier into the dc line. You may right-click on any of the rectifier-related fields and select [Quick Power Flow List](#) or [Bus View](#) from the local menu to view additional information about the rectifier bus.

Inv AC Number, Inv AC Name, Inv MW, Inv Mvar

Number and name of the inverter bus, and the real and reactive power flow from the inverter into the dc line. You may right-click on any of the inverter-related fields and select [Quick Power Flow List](#) or [Bus View](#) from the local menu to view additional information about the inverter bus.

Control Mode

Specifies how flow on the dc line is controlled. If this field is set to *Power*, then the line's MW flow is the control parameter. If the control mode is defined as *Current*, then the line's current is the control parameter.

Setpoint Magnitude, Setpoint Location

Specifies the initial value of the control parameter, which is expressed either in MW or in amps depending on the dc line's control mode. The Setpoint Location indicates if the setpoint should be set at the Inverter or Rectifier end of the DC line.

Set kV

Specifies the voltage of the dc line in kV.

Multi-Terminal DC Record Display

The Multi-terminal DC Line Display presents data describing each multi-terminal DC line in the case. The Multi-Terminal DC Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated records. The local menu also affords the opportunity to insert multi-terminal DC line records into the model or to delete existing ones.

To show the Multi-Terminal DC Line display, select **Network > DC Lines** in the [Model Explorer](#), and click on the Multi-terminal DC Lines tab of the transformer case information display.

The Multi-terminal DC Line display shows the following fields by default:

Number

The record number of each multi-terminal DC record.

Num Conv

The total number of converters in each multi-terminal DC network.

Num Buses

The total number of DC buses in each multi-terminal DC network.

Num Lines

The total number of DC lines in each multi-terminal DC network.

Mode

The control mode of the multi-terminal DC network: 0 is Blocked control, 1 is Power control, and 2 is Current control.

V. Cont. Bus

The number of the AC converter bus at which the DC voltage is controlled.

Switched Shunt Display

The Switched Shunt Display identifies all [switched shunt](#) devices in the case. Switched shunts are used in power systems either to inject additional MVR into the system (capacitive shunts) or to absorb excess reactive power (inductive shunts). They may also be used to regulate bus voltage within some specified range.

The Switched Shunt Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated switched shunts. You can find a specific switched shunt using its bus name or number, and you can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can insert a new shunt into the case or delete an existing shunt. You can also sort the switched shunt information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To call up the Switched Shunt Display, select **Network > Switched Shunts** in the [Model Explorer](#).

The Switched Shunt Display contains the following fields by default:

Number, Name

Number and name of terminal bus to which the switched shunt is attached. Note that multiple switched shunts are allowed at a bus, however only one switched shunt can have a Control Mode which is not fixed. Use the right menu options to inspect either the [Quick Power Flow List](#) or the [Bus View Display](#) for the terminal bus.

ID

Circuit identifier for the switched shunt device.

Reg Bus Num.

Number of the bus whose voltage is regulated by the switched shunt. If the regulation bus is the same as the terminal bus, this field is blank.

Status

The service status of the switched shunt. This field is toggleable.

Control Mode

Control Mode for the switched shunt. A switched shunt may operate either as *Fixed* if its reactive value is to be held constant, as *Discrete* if its reactive output is controlled automatically in discrete steps to regulate its terminal voltage, or as *Continuous* if its reactive injection is allowed to vary over a continuous spectrum of values to regulate its terminal voltage. A shunt will be switched either discretely or continuously to regulate its terminal bus' voltage between its high and low voltage limits provided the *Auto Shunt* field of its corresponding area is set to *Yes* and switched shunt control has not been disabled for the entire case. You may set the value of the *Auto Shunt* field from the [Area Display](#), and you can control case-wide enforcement of shunt switching from the Power Flow Solution tab of the [Simulator Options](#) Dialog. This Control Mode field is toggleable.

Regulates

Can be set to regulate either bus voltage or generator MVAR output.

Actual Mvar

The reactive power currently supplied by the switched shunt.

Volt High

Per unit high-voltage limit for the regulation range. It is important for discrete shunt control that Volt High exceed Volt Low by a nontrivial amount; otherwise, the output of the shunt may oscillate during the Power Flow Solution.

Volt Low

Per unit low-voltage limit for the regulation range.

Reg Volt

Actual per unit voltage at the regulated bus. When Control Mode is either *Discrete* or *Continuous*, this voltage should be between Volt Low and Volt High.

Deviation

Deviation of the regulated bus' actual per unit voltage from the desired regulation voltage. If the actual voltage is within the regulation range, this field is zero. If the voltage is greater than Volt High, the deviation is positive. It is negative if the actual voltage is less than Volt Low.

Nominal Mvar

The Nominal Mvar field gives the initial amount of reactive power the device would supply (in Mvars) if its terminal voltage were 1.0 per unit.

Max Mvar, Min Mvar

The maximum and minimum Mvar range for the switched shunt.

Interface Display

The Interface Display is used to show the net real power (MW) flow on a group consisting of one or more of the following devices: 1) transmission lines (AC and DC) and/or transformers, 2) total tie-lines between two adjacent areas, 3) total tie-lines between two adjacent zones, 4) Generators and Loads, 5) Injection Groups, 6) multi-section lines, 7) other interfaces and 8) Contingency Actions . Interface information is useful because secure power system operation often requires that the flow on such groups be less than some limit value. For example, interface information could be used as "proxies" for other types of security constraints, such as voltage or transient stability limitations. Another major potential use for interfaces is in the [Optimal Power Flow](#) solution to monitor flows on groups of devices, such as a set of transmission lines, generation leaving a plant, load demanded in a certain region, etc. Interface information can also be extremely useful for summarizing the flows occurring on a large network. Interface flows can be monitored using the [Limit Monitoring Settings Display](#).

The Interface Display presents more detailed information for each interface that has been defined for the case. The Interface Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated interfaces. The local menu also affords the opportunity to insert new interface definitions either singly (**Insert**) or as a group ([Automatic Insertion](#)) into the model or to delete existing ones. In addition, PowerWorld has also added options for reading NERC flowgate files ([Load NERC Flowgates](#)) and writing NERC flowgate files ([Save NERC Flowgates](#)) to the local menu.

You can sort the interface information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Interface Display according to the constraints of the [area/zone/owner filters](#).

To show the interface display, select **Aggregations > Interfaces** in the [Model Explorer](#).

Interface information can be saved in the "*.aux" auxiliary file format. See [Auxiliary Files](#) for details.

The interface Display is divided into two tabs: **Interface** and **Interface Elements**.

Interface Elements Tab

The **Interfaces Elements tab** has a list of tabs which lists each types of element that can belong to an interface. There is a check-box below in the middle of the display which allows you to **Only show elements that belong to at least one interface**. As you select an element in the top half of the display, the bottom half of the display will list each interface which contains the selected element. Again, these provide a nice way to look through the elements that make up each interface.

Interface Tab

The **Interfaces tab** has a list of interfaces at the top of the display. When on the Interface tab, in the bottom half of the tab is a set of subtabs which display all the elements of the presently selected interface from the top half of the display. This set of sub tabs starts with the **Elements** tab which lists all the elements of the interface regardless of the type of element. The remainder of the tabs each lists the interface elements of a specific type of interface element such as Branches, Branch Open/Close, DC Lines, Generators, Loads, Injection Groups, Multi-Section Lines and other interfaces. These subtabs provide a nice way to look through the elements that make up each interface. The top half of the Interface display contains the following fields:

Number

Numeric identifier for the interface.

Name

Alphanumeric identifier for the interface (24 characters maximum).

Interface MW

Current MW flow on the interface. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

MW Limit

Current rating for the interface in MW.

Percent

The actual MW flow on the interface as a percentage of the MW limit.

Monitor Direction

The current direction in which the MW flow is being monitored. The possibilities are From - To or To - From.

Monitor Both

Toggle this field to Yes in order to monitor the flow on the interface in both directions, instead of a single direction as designated by the Monitor Direction field.

Lim MW A, Lim MW B, Lim MW C

These three fields display the values of the three possible limits for the interface.

Has Contingency

This is a Yes or No field that will be set to Yes if any of the elements making up part of the interface are at a device limit.

Contingent MW Flow

If the interface defined contains a contingency action, the Contingent MW Flow is the approximated flow amount that would be added to the interface should the contingency occur.

Base MW Flow

The base MW flow is the flow on the interface prior to any considered contingency elements.

Nomogram Display

The Nomogram Display is used to modify and create Nomogram definitions. Nomograms are combinations of two [interfaces](#) for monitoring combined flows on the interfaces concurrently. These Nomogram interfaces will have a limit definition that defines a region of allowed flow on the interfaces, and can be monitored in many tools in Simulator as potential constraints, such as the contingency analysis reporting of interface violations. Another major potential use for nomograms is in the [Optimal Power Flow](#) solution to monitor the flow on a pair of interfaces. Nomogram information can also be extremely useful for summarizing the flows on a pair of interfaces whose operation and allowed flow are closely tied together. Nomogram flows can be monitored using the [Limit Monitoring Settings Display](#).

The Nomogram Display presents more detailed information for each nomogram that has been defined for the case. The Nomogram Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated nomograms.

You can sort the nomogram information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Interface Display according to the constraints of the [area/zone/owner filters](#).

To show the interface display, select **Aggregations > Nomograms** in the Model Explorer.

Nomogram information can be saved in the "*.aux" auxiliary file format. See [Auxiliary Files](#) for details.

The Nomogram display contains the following fields:

Name

Alphanumeric identifier for the nomogram.

Int A Flow

Current MW flow on the first interface of the interface pair forming the nomogram. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

Int B Flow

Current MW flow on the second interface of the interface pair forming the nomogram. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

% Limit, Max Nomo-interface

The actual MW flow on the nomogram as a percentage of the combined nomogram limit.

Monitor

Indicates whether or not the nomogram should be monitored either for violations or as a constraint in the [Optimal Power Flow](#).

Limit Group

The name of the limit group the nomogram is a member of for [limit monitoring](#).

Injection Group Display

The Injection Group Records Display presents data describing each injection group in the case. This display is available by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). This display is organized into two tabs with various sections available under each of the tabs. The tabs and sections provide options for displaying information about injection groups and their associated participation points in a manner in which it is most useful to the user. Injection groups can be displayed with their associated participation points or participation points can be displayed with the injection group to which they belong.

Injection Groups Tab

This tab is organized into a top and bottom section. The top section contains the Injection Groups [Case Information Display](#) that lists all of the defined injection groups. The [local menu](#) is available by right-clicking on the display and contains options available for most case information displays as well as the option to **Show Dialog** to view the [Injection Group Dialog](#).

By default, the Injection Groups Case Information Display contains the following fields:

Name

The name of the injection group. To change the name of an injection group, simply type a new name in the corresponding cell.

Number of Gens

Identifies the number of generators contained in the injection group.

% MW Gen ParFac

Indicates the degree to which generators will contribute to the MW output of the injection group relative to loads. An injection group that has a % MW Gen ParFac value of 100% receives all of its output from generator points; an injection group that has a % MW Gen ParFac of 50% and a % MW Load ParFac of 50% receives equal MW contributions from its constituent loads and generators.

Number of Loads

Identifies the number of loads contained in the injection group.

% MW Load ParFac

Indicates the degree to which MW loads will contribute to the MW output of the injection group relative to generators. An injection group that has a % MW Load ParFac value of 100% receives all of its output from load points; an injection group that has a % MW Load ParFac of 50% and a % MW Gen ParFac of 50% receives equal MW contributions from its constituent loads and generators.

% Mvar Load ParFac

Indicates the degree to which Mvar loads will contribute to the Mvar output of the injection group relative to switched shunts. An injection group that has a % Mvar Load ParFac value of 100% receives all of its Mvar output from load points; an injection group that has a % Mvar Load ParFac of 50% and a % Mvar Shunt ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Number of Shunts

Identifies the number of switched shunts contained in the injection group.

% Mvar Shunt ParFac

Indicates the degree to which switched shunts will contribute to the Mvar output of the injection group relative to Mvar loads. An injection group that has a % Mvar Shunt ParFac value of 100% receives all of its Mvar output from switched shunt points; an injection group that has a % Mvar Shunt ParFac of 50% and a % Mvar Load ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Total MW Injection

Indicates the current total MW injection of all elements in the injection group.

Total Mvar Injection

Indicates the current total Mvar injection of all elements in the injection group.

The bottom section contains additional tabs: **Participation Points (All)**, **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. These tabs contain tables, which are variations of the [Participation Point Records Display](#), that list all of the individual participation points for the selected injection group by element type. Participation points can be added, deleted, or modified for the selected injection group from any of the tables on these tabs. The **Participation Points (All)** table lists only the fields that are relevant for defining participation points while the specific element tables also make available the fields that are specific to that element type.

The **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups** tabs have the option to **Show points that are contained by other injection group points**. This option is only relevant if an injection group contains a participation point that is another injection group. When this option is checked, the individual participation points that are contained in the injection group point will also be displayed. The **Contained by** field will list the name of the injection group that contains a particular participation point. When this option is not checked, the points displayed will be points that are contained in the selected injection group only and no additional information will be provided about the points contained inside any injection group points belonging to the selected injection group. If the **Contained by** field is empty, this means that the participation point is explicitly defined with the selected injection group and does not belong to it through an injection group point.

Participation Points Tab

This tab is organized into a top and bottom section. The top section contains tabs for **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. Tables on these tabs will list power system elements of each particular type. If the option to **Only show elements that belong to at least one injection group** is not checked, then all elements of each type in the case will be listed on one of these tabs. If this option is checked, then only those elements that belong to an injection group will be listed. The tables on these tabs are [Case Information Displays](#) for each particular type and have the full functionality of case information displays.

The bottom section is an Injection Groups Case Information Display. When an element from one of the tables in the top section of the tab is selected, this table will be populated with the list of injection groups to which the selected element belongs.

Injection Group Dialog

The Injection Group Dialog provides information about injection groups and allows their modification. Specifically, the Injection Group Dialog lists the number of generators, loads, and switched shunts contained in the group and the percentage contribution of generators, loads, and switched shunts to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which points can be added and deleted from the injection group's list of participants and the various attributes of the points can be changed.

To view the Injection Group Dialog for a particular injection group, open the [Injection Group Display](#) by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). Find the injection group of interest in the Injection Groups Case Information Display and right-click on it. Then select **Show Dialog** from the resulting local menu.

The Injection Group Dialog contains the following fields and controls:

Name

Identifies the injection group whose information is currently displayed. Selecting a different name from this dropdown box will display information for another injection group.

New, Delete

To insert a new injection group from this dialog, press **New**, and supply the name for the new injection group. To delete the injection group that is currently being shown, click **Delete**.

Save

Changes made to injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click the **Save** button to store any updates with the case.

Gens

Displays the number of generators contained in the injection group.

Loads

Displays the number of loads contained in the injection group.

Shunts

Displays the number of switched shunts contained in the injection group.

% MW Gen Part., % MW Load Part., % MVR Load Part., % MVR Shunt Part.

Displays the relative contributions of generators, loads, and switched shunts to the output of the injection group.

OK, Cancel

Changes made to the injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click **Cancel** to close the dialog without saving the changes. Click **OK** to close the dialog and save the changes.

Participation Points

The tab on the bottom of the Injection Group Dialog lists the points that make up the injection group. This display is called the [Participation Point Records Display](#). By right-clicking on this display, points can be added and deleted from the injection group. Properties of specific points can be changed by

entering changes directly in the table. The **Insert Points** button will open the [Add Participation Points Dialog](#) that can also be used for modifying the points.

Participation Point Records Display

The Participation Point Records Display shows information about the points that comprise a particular injection group. This display can be accessed for a particular injection group from the [Injection Group Dialog](#) or the [Injection Group Display](#).

The Participation Point Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which participation points can be deleted or added to the injection group through the [Add Participation Points Dialog](#). Selecting **Insert** from the local menu will open the Add Participation Points Dialog that will allow the addition, deletion, and modification of participation points in the injection group.

The Participation Point Records Display shows the following fields by default:

Point Type

Every participation point is a generator (GEN), load (LOAD), switched shunt (SHUNT), or injection group (INJECTIONGROUP).

Number

Identifies the number of the bus to which a generator, load, or switched shunt is connected.

Name

Identifies the name of the bus to which a generator, load, or switched shunt is connected.

ID

Identifies the ID of the generator, load, or switched shunt or the name of the injection group.

AutoCalc?

If the value of **AutoCalc** is *YES*, the participation factor of the point is re-calculated with every use to be consistent with the way the point's participation factor was initially defined. If the value of **AutoCalc** is *NO*, the participation factor is assumed fixed at its present value.

Initial Value

Indicates how the participation factor of the point was originally computed. For generators, the possible values of this field are

SPECIFIED The participation factor was specified as a constant.

MAX GEN INC The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

MAX GEN DEC The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

MAX GEN MW The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

SPECIFIED The participation factor was specified as a constant.

LOAD MW The participation factor was defined as the size of the load in MW.

For switched shunts, the Initial Value property can assume four possible values:

SPECIFIED The participation factor was specified as a constant.

MAX SHUNT INC The participation factor was defined as the difference between the maximum Mvar output of the switched shunt and its present nominal Mvar output.

MAX SHUNT DEC The participation factor was defined as the difference between the present nominal Mvar output and the minimum Mvar output of the switched shunt.

MAX SHUNT MVAR The participation factor was defined as the maximum Mvar output of the switched shunt.

For injection groups, the Initial Value property can assume only one possible value:

SPECIFIED The participation factor was specified as a constant.

The **Initial Value** field is important if saving the injection groups to an auxiliary file and using them with another case that might have a different generation dispatch or load profile. If the **Initial Value** for a point is specified as *GEN MAX INC*, for example, and the point is loaded from an auxiliary file into another case, Simulator will re-calculate the point's participation factor to match the generator's positive MW reserve in that case.

The **Initial Value** field is also important if the **AutoCalc** field is set to *YES*, because **AutoCalc** uses the rule defined by the **Initial Value** field to recalculate the participation factor with every use. For example, if the **Initial Value** is *MAX GEN INC* and **AutoCalc** is *YES*, the point's participation factor will be updated to match the generator's MW reserve every time the point is accessed.

When the value of the **Initial Value** field is toggled, the point's participation factor, shown in the **ParFac** field, will update to match the new definition. The **Initial Value** field must equal *SPECIFIED* to be allowed to change the value of the point's **ParFac** field by typing the new value directly into the field.

If **AutoCalc** is *NO* and the injection group is not intended to be used with any other case, then the **Initial Value** field should either be ignored or set to *SPECIFIED*.

ParFac

Indicates the participation factor of the participation point. The participation factor defines the relative contribution of the point to the total output of the injection group. The load, generation, or switched shunt change associated with each point is calculated based on the value of the participation factor, with values for points having the largest participation factors experiencing the greatest change.

The participation factor for injection group participation points indicates the relative contribution of the entire injection group to the total output of the injection group to which it belongs. The participation factors of the individual points in the injection group point further define the relative contribution of these points to the total output of the injection group. The following two injection groups provide an example of how injection group points impact the injection group to which they belong when the points are normalized for use:

<u>Injection Group 1</u>	<u>Injection Group 2</u>
Gen 1, ParFac = 50	Gen 5, ParFac = 40
Gen 2, ParFac = 30	Load 1, ParFac = 20
InjectionGroup 2, ParFac = 20	Load 2, ParFac = 20
Total = 100	Total = 80

When Injection Group 1 is used, the participation factors are normalized so that each element provides the following contribution:

Gen 1, Normalized ParFac = $50/100 = 0.5$
 Gen 2, Normalized ParFac = $30/100 = 0.3$
 Gen 5, Normalized ParFac = $20/100 * 40/80 = 0.1$

Case Information Displays

Load 1, Normalized ParFac = $20/100 * 20/80 = 0.05$

Load 2, Normalized ParFac = $20/100 * 20/80 = 0.05$

Total Normalized = 1.0

To add points to the injection group, right-click on the Participation Point Records Display and select **Insert** from the local menu. This opens the [Add Participation Points Dialog](#).

Island Display

The Island Display presents information on the system's islands. An island consists of a group of buses that are electrically connected via ac transmission lines and transformers and thus operate in synchronism with one another. Multiple islands can be connected together via dc transmission lines. Each island requires a slack bus. All systems have at least one island, which may encompass the entire system.

The Island Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information. You can also sort the islands information by clicking on the heading of the field by which you want to sort.

To show this display select **Aggregations > Islands** in the [Model Explorer](#). The display is only available in run mode.

You cannot modify any fields on this display.

The display has the following fields for each island:

Slack Bus Number, Slack Bus Name, Slack Bus Area

Number, name and area of the slack bus for the island. Each island requires at least one slack bus.

Total Buses

Total number of buses in the island.

Energized

Indicates whether the island is connected to a source of power.

Gen MW, Gen Mvar

Total real and reactive generation for the island.

Load MW, Load Mvar

Total real and reactive load for the island.

Scheduled Exports

The power scheduled to be provided by the island to other regions of the system. Because the island is isolated from the rest of the system, this export requirement is currently not met.

DC MW Exports

The power provided by the island to other regions of the system across a DC line.

MW Transactions Display

The MW Transactions Display is a quick way to view the defined base transactions between areas within the load flow case.

The **Matrix of Transactions** display is set up as a matrix of transactions, with the areas listed as both the column and row identifiers. It is important to note that the direction of the transaction is such that the area represented by the **row** is assumed the exporting area, and the area represented by the **column** is assumed the importing area. For example, if there is a 50 MW transaction from area one to area two, you will see a +50 in the matrix in row 1, column 2. However, if you look at row 2, column 1, you will see a – 50. This is because the grid is displaying the EXPORT from area two to area one, but since area two is importing, not exporting, the value is represented as negative.

Each row and column position, with the exception of the diagonal positions (it does not make sense for an area to export to itself), can be directly modified by the user in this information display. As you type a value in one of the matrix positions, Simulator automatically fills the symmetric matrix position with the negative of the value you enter. This makes for a quick and easy location for adding and removing transactions from the case.

Note that you cannot modify **Unspecified** transactions in the MW Transactions display. This makes sense, as the unspecified transactions have only one associated area. To modify unspecified transaction amounts, you must open the [Area Information Dialog](#) for the area you wish to modify, and change the Unspecified transaction amount.

The **List of Transactions** is just another way of showing the transactions between areas. The List of Transactions Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated areas. You can also sort the area records by clicking on the heading of the field by which you want to sort.

Note that a special feature is available in the right-click local menu called **Clear Transactions and auto-insert tieline transactions**. By choosing this option all MW transactions in the case will be deleted, and all Unspecified MW transactions for each area will be set to zero. Then new MW transactions will be created between each pair of areas that area directly connect to one another. The amount of the new MW transactions will be set equal to the actual sum of the flow on the tielines between the connected areas.

By default, the List of Transactions display contains the following fields:

Export Area Number, Export Area Name

Exporting area number and alphanumeric identifier.

Other Area Number, Other Area Name

Importing area number and alphanumeric identifier.

MW Transfer

Value of the transfer in MW.

Transaction Dialog

The Transaction Dialog can be used to modify or create Base Interchange Transactions between two areas. This dialog can be opened by right-clicking in the Base Interchange table of the [Area Information Dialog](#) and choosing Show Dialog to see an existing transaction definition, or Insert to add a new transaction.

The transaction dialog is divided into two pages of controls:

Information

Transacting Area

This is the "from" area for the transaction. For an export from this area, the transaction value will be positive. For an import into the Exporting Area, the transaction value specified would be negative. Flow out (export) of the exporting area is always considered positive.

Transaction To Area

This is the "to" area for the transaction.

Transaction ID

New in Simulator version 10 is the ability to have multiple transactions defined between the same two areas. Because of this, it is now required that transactions also have a Transaction ID.

Rename Transaction ID

If you wish to change the transaction ID for a particular transaction, enter the new value in the **Transaction ID** field, and press this button.

Switch Directions

Press this button if you wish to reverse the defined Exporter and Importer for the transaction.

Transaction MW Amount

The MW amount of the transaction being defined. This value should be positive for an export from the Exporting area to the Importing area. The value can also be entered as negative to define a transaction into the Exporting area from the Importing area.

Transaction Minimum MW

The minimum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Transaction Maximum MW

The maximum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Exports/Imports Transmission Charge

The cost to transfer power, in \$/MWh. This adds an economic penalty for making the transfer, making the transfer less likely to take place. Half the charge is assigned to the buyer and half to the seller.

Transaction Enabled

Transaction can now be defined and either enabled or disabled. Any disabled transactions will be ignored in both a standard power flow solution and an OPF solution.

Transaction Dispatchable in OPF

Checking this box enables the transfer to be dispatched by the OPF algorithm. Dispatching the transaction makes the two areas of the transaction appear to be one area for the purpose of economically dispatching the generation in the two areas. The transaction can have a maximum and minimum transfer amount when dispatchable, and a transmission charge associated with the transaction.

Determine Price in OPF

Checking this box allows the OPF algorithm to determine the cost associated with the transfer. The cost is determined by the marginal cost of enforcing the power balance constraint for the combined areas. This is the typical way to implement a transfer if *both* areas are on OPF control. If only one of the two areas are on OPF control, then the area which is off of OPF control needs to specify a price for the transfer. This is done by explicitly defining a piecewise linear cost curve.

Piecewise Linear Transaction Cost Curve

These two curves are only enabled if the option **Transaction is Dispatchable in OPF** is checked and the option **Determine Price in OPF** is unchecked. These two curves can be defined for the purpose of assigning a price to the transfer of power between one area on OPF control and another area which is not on OPF control. Separate curves can be defined for export transactions (from the Exporter to the Importer) and import transactions. To add points to the curves, simply right-click in the grid and choose Insert from the local menu. Enter the MW value and corresponding marginal cost for the inserted breakpoint of the piecewise linear curve you are defining. To delete a point, right-click on that row in the grid and choose Delete from the local menu.

Custom

The Custom page of the Transaction dialog is simply a location to log information about the transaction in the Memo box, and to see the values (if any) stored in some of the custom fields for the transaction. To log information about the transaction, simply switch to the Custom page on the dialog, and start typing your information or comments about the transaction in the page, or enter custom values in one of the custom fields.

Limit Monitoring Settings

Simulator offers many tools to study the capabilities of a power system. Examples include:

- [Contingency Analysis](#)
- [Available Transfer Capability \(ATC\)](#)
- [Optimal Power Flow \(OPF\)](#)
- [Security Constrained OPF \(SCOPF\) Overview](#)
- [PV Curve and QV Curve Tool](#)

All of these tools make extensive use of power system limits. Limits for various power system elements include:

- MVA (or Amp) limits on transmission lines and transformers
- MW limits on Interfaces
- High and low voltage limits for Buses

The accuracy of all these limits is very important, as is the specification of which limits should be monitored. While ensuring the accuracy of input data such as power system limits must be left to the user, PowerWorld Simulator provides several ways to specify which limits should be monitored. Limit Monitoring is specified according to settings for the Area, Zone and Limit Group to which the power system element belongs. A power system element is monitored only if ALL of the following conditions are met.

Conditions for Monitoring an Element's Limit

- Its Monitor field is set to **YES**
- Its Limit Group is Enabled
- Its Area is set to Report Limits and it meets the KV range for reporting
- Its Zone is set to Report Limits and it meets the KV range for reporting

The [Limit Monitoring Settings Dialog](#) gives you ability to specify all these settings to setup the limits you want to monitor, and thus enforce in the various tools that Simulator provides.

Limit Monitoring Settings and Limit Violations Dialog

The Limit Monitoring Settings Dialog gives you ability to specify which limits you want to monitor, and thus enforce in the various tools that Simulator provides. In general, keep in mind that a bus, transmission line/transformer, or interface's limit is monitored only if the following conditions are met.

Conditions for Monitoring an Element's Limit

- Its Monitor field is set to **YES**
- Its Limit Group is Enabled
- Its Area is set to Report Limits and it meets the KV range for reporting
- Its Zone is set to Report Limits and it meets the KV range for reporting

The following fields are shown on the display:

Elements To Show

Change this value to modify which elements are displayed in the Buses, Lines, and Interfaces. Set it to...

All Elements to show all Buses, Lines and Interfaces regardless of monitoring settings.

Monitored Elements to only display Buses, Lines and Interfaces that meet the conditions for monitoring. See [Limit Monitoring Settings](#) for more information on setting which values are monitored.

Violating Elements to only display Buses, Lines and Interfaces that are violating. See [Limit Monitoring Settings](#) for more information on setting what is considered a violation.

Number of Violations

High Voltage Buses and Low Voltage Buses

Shows the total number of bus voltage magnitude violations. These violations are shown on the **Bus Voltage Magnitude Limit Violations Display**. This field is read-only.

Low Voltage Suspects

Shows the total number of buses whose voltages have fallen below a designated threshold to indicate a low voltage solution is being reached in some location in the system.

Line/Transformer Violations

Shows the total number of violations reported on **Line and Transformer Limit Display**. This field is read-only.

Interface Violations

Indicates the total number of interface violations. These violations are shown on the **Interface Violations Display**. This field is read-only.

Limit Group Values

This part of the dialog shows information about the Limit Groups with greater description shown below.

Save/Load Monitoring Settings

The Save and Load Monitoring Settings buttons allow the user to save the current set or load a previously saved set of Limit Monitoring Settings options. The options are saved in a [PowerWorld Auxiliary File](#) text format.

Do not monitor radial lines and buses

Check this box to ignore limits on radial lines and buses throughout the case. In this instance a radial bus is defined as a bus that is connected to the rest of the power system by a single inservice transmission line. A radial line is defined as a line connected to a radial bus. This means that in a 5 lines in series that connect to a radial load, only the last bus and last line will be considered radial.

If you would like to better determine which lines are "radial" in a more traditional sense, use the feature [Branches that Create Islands](#) that allows you to easily generate a list of branches that if taken out of service will split an existing island into two. These represent lines which may be considered radial.

Tabs on the Limit Monitoring Dialog

The tabs on the Limit Monitoring Settings Dialog allow you to change these settings. These tabs contain tables showing lists of the respective elements. Since these tables are another variety of the [Case Information Displays](#), you may interact with it in a familiar manner. Click on any of the field headings to sort by that field. Right-click on the display to call up the display's [local menu](#). From the local menu, you can print the violations, [copy the violation records](#) to the Windows clipboard for use with another application, modify the format and content of the violations listing, view the information dialog of the respective element, and view the [Quick Power Flow List](#) or [Bus View Display](#).

Buses Tab

Each tab shows a list of the respective type of power system element. The important columns include

Number, Name Bus number between 1 and 99,999, and its alphanumeric identifier.

Area Name Alphanumeric identifier of the bus' area.

Monitor : toggle this between **YES** and **NO** to set whether the specific element should be monitored.

Limit Group : toggle this value to specify which Limit Groups the element belongs to

PU Volt Bus' per unit voltage magnitude.

Volt (kV) Bus' actual voltage magnitude in kV. This is the per unit voltage magnitude multiplied by the bus' nominal voltage.

Limit Low PU Volt Bus' individual low voltage limit used when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the low per unit voltage limit specified in one of the eight low voltage limit sets that can be defined with the bus. Which voltage limit set's value is displayed depends on which limit set is specified as the "Bus Low Rate Set" in the limit group to which the bus is assigned.

Limit High PU Volt Bus' individual high voltage limit used when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the high per unit voltage limit specified in one of the eight high voltage limit sets that can be defined with the bus. Which voltage limit set's value is displayed depends on which limit set is specified as the "Bus High Rate Set" in the limit group to which the bus is assigned.

Contingency Limit Low PU Volt Bus' individual low voltage limit used during a contingency when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the

bus is set to YES. The value will then reflect the contingency low per unit voltage limit specified in one of the eight contingency low voltage limit sets that can be defined with the bus. Which contingency voltage limit set's value is displayed depends on which limit set is specified as the "Bus CTG Low Rate Set" in the limit group to which the bus is assigned.

Contingency Limit High PU Volt Bus' individual high voltage limit used during a contingency when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the contingency high per unit voltage limit specified in one of the eight contingency high voltage limit sets that can be defined with the bus. Which contingency voltage limit set's value is displayed depends on which limit set is specified as the "Bus CTG High Rate Set" in the limit group to which the bus is assigned.

Note: If Elements to Show is set to Monitored Elements, then only elements which meet the conditions for monitoring will be displayed in these lists.

Lines Tab

Each tab shows a list of the respective type of power system element. The important columns include

From Bus Number and Name "From" bus number and name. For transformers, the from bus is the tapped side. Right-clicking on either of these fields allows you to see all the flows measured at the "from" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

To Bus Number and Name "To" bus number and name. For transformers, the to bus is the untapped side. Right-clicking on either of these fields allows you to see all the flows into the "to" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

Circuit Two-character identifier used to distinguish between multiple lines joining the same two buses.

Monitor : toggle this between **YES** and **NO** to set whether the specific element should be monitored.

Limit Group : toggle this value to specify which Limit Groups the element belongs to

Used Limiting Flow, Limit, Used % of Limit The flow at the end of the branch selected for measurement, and its MVA or Amp limit. The percentage equivalent of the flow to its limit is given in the Used % of Limit column.

MVA or Amps? Units used with the Max Flow and the Limit field. All flows are expressed either in MVA or amps.

Note: If Elements to Show is set to Monitored Elements, then only elements which meet the conditions for monitoring will be displayed in these lists.

Interfaces and Nomograms Tab

Similar to the Buses and Lines Tabs shown above. The important columns are the **Monitor** and **Limit Group** columns

Area Reporting and Zone Reporting Tabs

These tabs display all the Areas and Zones in the system. The important columns are

Report Limits: toggle this between **YES** and **NO** to set whether the specific element should be monitored.

Report Min kV and Report Max kV: Only buses and lines within this kV range will be monitored.

Modify/Create Limit Group Tab

Every power system element belongs to a single Limit Group. By default, all power system elements are in the same Limit Group that is named "**Default**". New limit groups can be added by right-clicking in the table and choosing **Insert** from the popup menu. Additional limit groups give the flexibility of assigning devices to different groups, where each group can have its own set of defined limit information.

The limit group stores important values regarding the enforcement and monitoring of the power system elements within the Limit Group. The values are

Branch Percentage: The percentage to which Simulator's study tools will limit a line or transformer. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), then lines will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the line below this percentage.

Line Rate Set: You may define eight different ratings to transmission lines or transformers (for more information see [Line/Transformer Information](#)). Change this value to specify which rating set should be used for lines/transformers in the limit group.

Amps or MVA: Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the Treat Line Limits As Equivalent Amps is checked, the limits for transmission lines are reported in amps rather than MVA. If this box is not checked, limits for both transmission lines and transformers are expressed in MVA.

For reference, note that the amp rating of a line is derived from the MVA rating using the formula

$$\text{Amp Rating} = \frac{\text{MVA Rating} \times 1000}{\sqrt{3} \times \text{Line Voltage}}$$

Interface Percentage: The percentage to which Simulator's study tools will limit an interface. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), then interfaces will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the interface below this percentage.

Interface Rate Set: You may define eight different ratings to an interface. Change this value to specify which rating set should be used for interfaces in the limit group.

Nomogram Percentage: The percentage to which Simulator's study tools will limit a nomogram.

Low PU Volt: Buses will be flagged as violated if they fall below this per unit voltage.

High PU Volt: Buses will be flagged as violated if they go above this per unit voltage.

Disabled: Set the value to **YES** to ignore all power system element limits in the Limit Group. Set the value to **NO** to monitor limits according to the settings of the Limit Group.

Contingency Line Rate Set: This field specifies the rating set used for post-contingency monitoring of Lines/Transformers.

Interface Contingency Rate Set: This field specifies the rating set used for post-contingency monitoring of Interfaces during Contingency Analysis.

Limiting End: Specified as higher or lower, this field determines whether the higher or the lower flow amount on the element is used for reporting a limit violation. If you use higher, it is possible that the lower flow is not violating the line limit. If you use lower, then you are guaranteed that the element limit is being violated at both ends of the element.

Use Limit Cost: If this field is set to yes, you are enabling the capability to have a cost function associated with enforcing constraints. This cost function can be viewed as similar to a generator cost function, in that as the constraint becomes overloaded by larger amounts,

the marginal cost of enforcing the constraint will increase. For setting the piecewise linear limit cost curve, right-click and choose show dialog from the popup menu to access the Limit Group Dialog.

Often monitoring elements according to Area and/or Zone is all that you need. However, if you need to monitor specific groups of power system elements and not others then you need to create some new Limit Groups.

Limit Group Dialog

The Limit Group Dialog reflects much of the same information that is summarized in the [Limit Monitoring Settings](#) Dialog's Modify/Create Limit Groups page. The same information can be changed here as on the previously mentioned display. One important feature that is unique to this dialog is the ability to define a piecewise linear limit cost curve, which can be used for setting up "soft" constraints used in the [Security Constrained Optimal Power Flow](#).

Disabled

Check this option to ignore all power system element limits in the Limit Group. Leave the box unchecked to monitor limits according to the settings of the Limit Group.

Add New Limit Group

Clicking this button will create a new limit group, displaying a dialog in which the name for the new limit group can be specified.

Rename Limit Group

Clicking this button will display a dialog in which a new name for the current limit group can be specified.

Lines/Interfaces

% of Limit for Reporting (Branches)

The percentage to which Simulator's study tools will limit a line or transformer. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), lines will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the line below this percentage.

Line/Transformer Rating Set

You may define eight different ratings to transmission lines or transformers (for more information see [Line/Transformer Information](#)). Change this value to specify which rating set should be used for lines/transformers in the limit group.

Line/Transformer Contingency Rating Set

This field specifies the rating set used for post-contingency monitoring of Lines/Transformers.

Treat Transmission Line Limits as Equivalent Amps

Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the Treat Line Limits As Equivalent Amps is checked, the limits for transmission lines are reported in amps rather than MVA. If this box is not checked, limits for both transmission lines and transformers are expressed in MVA.

For reference, note that the amp rating of a line is derived from the MVA rating using the formula

$$\text{Amp Rating} = \frac{\text{MVA Rating} \times 1000}{\sqrt{3} \times \text{Voltage}}$$

Limiting End of Line

Specified as higher or lower, this field determines whether the higher or the lower flow amount on the element is used for reporting a limit violation. If you use higher, it is possible that the lower flow is not violating the line limit. If you use lower, then you are guaranteed that the element limit is being violated at both ends of the element.

% of Limit for Reporting (Interfaces)

The percentage to which Simulator's study tools will limit an interface. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), interfaces will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the interface below this percentage.

Interface Rating Set

You may define eight different ratings to an interface. Change this value to specify which rating set should be used for interfaces in the limit group.

Interface Contingency Rating Set

This field specifies the rating set used for post-contingency monitoring of Interfaces during Contingency Analysis.

% of Limit for Reporting (Nomograms)

The percentage to which Simulator's study tools will limit a nomogram.

Use Limit Cost

If this field is checked, you are enabling the capability to have a cost function associated with enforcing constraints. This cost function can be viewed as similar to a generator cost function, in that as the constraint becomes overloaded by larger amounts, the marginal cost of enforcing the constraint will increase.

Once the box is checked, the table for defining the limit cost function will become enabled. The starting point (% Flow) must be at or above 100%. You can then begin inserting additional points in the piecewise linear curve by right-clicking in the table and selecting Insert Point from the popup menu, followed by entering the new percent flow and marginal cost. Note that the cost function must be strictly increasing, meaning the next marginal cost value must be equal to or greater than the immediately previous value.

When constraints are given the ability to use this limit cost curve, it effectively gives the optimal power flow the ability to "dispatch" the limit of the elements according to the marginal costs of the limit cost curve. You are determining the point on the limit cost curve where the shadow price of enforcing the constraint is met. Thus the OPF routine will determine how much you will allow the element to be overloaded by giving the element the piecewise linear curve. The OPF will solve the problem, and optimize the constraints as it determines shadow prices for each constraint and adjusts the limit accordingly during the iterations of the routine. Since these are somewhat flexible limit assignments to the elements, they are sometimes considered "soft" constraints.

Buses/Summary

Low Per Unit Limit

Buses will be flagged as violated if they fall below this per unit voltage.

High Per Unit Limit

Buses will be flagged as violated if they fall below this per unit voltage.

Low-voltage solution flag

Enter a per unit value at which you wish to measure a voltage and consider the solution to be a low voltage solution.

Include Out-Of-Service Buses

Check this check-box to include out-of-service buses.

Elements using Limit Group

Lists the number of elements that are members of the selected limit group.

Owner Data Information Display

The Owner Data display will display the owners sorted by owner number, and show the total number of devices, as well as a breakdown of the number of specific devices. The owner display will also summarize the total device output or demand for the owned devices, such as generator output and load demand.

You can right click on the owner grid to open the local menu, from which you can perform various actions and view an informational regarding the owner selected in the grid. Once an [owner dialog](#) is displayed for a particular owner, you can browse through information on all the owners.

Owner Dialog

The Owner Dialog displays summary information about the devices designated as owned by the selected owner.

The dialog for an individual owner displays each of the elements for that owner, along with the percentage of ownership, in the device pages labeled [Buses](#), [Loads](#), [Generators](#), and [Lines](#).

The dialog also contains a [memo](#) page for making comments or notes about the selected owner.

General Info

Load and Generation

The information in the Load and Generation section provides a summary of the total injections of the owned devices. This includes a total of all the load, generation, and shunt injections owned.

Summary of Owner Objects

This information section simply lists a total number of all owned devices. Note that buses and loads do not currently have fractional ownership. However, generators and transmission lines do have the capability to be partially owned by more than one owner. Therefore it is possible to see fractional ownership amounts for the number of generators or lines.

Generator Costs and OPF Results

This information provides a summary of LMP and cost information determined by running an [Optimal Power Flow](#) solution. The information provided is determined only for devices owned by the selected owner.

Owned Bus Records Display

The Owned Bus Records display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Bus Display](#). In addition to the information available for viewing from the [Bus Display](#), the Owned Bus Records Display also shows the owner number and percentage of ownership. Currently, Simulator only allows one owner per bus, so by default the ownership percent is always 100%.

Owned Load Records Display

The Owned Load Records display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Load Display](#). In addition to the information available for viewing from the [Load Display](#), the Owned Load Records Display also shows the owner number and percentage of ownership. Currently, Simulator only allows one owner per load, so by default the ownership percentage is always 100%.

Owned Generator Records Display

The Owned Generator Records Display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Generator Display](#). In addition to the information available for viewing from the [Generator Display](#), the Owned Generator Records Display also shows the owner number and percentage of ownership. Currently, Simulator allows up to four owners for one generator, with a total percentage ownership between 0 and 100%. To make changes to the owners and ownership percentages for a generator, right-click on the grid and choose Show Dialog and select the [Owners, Area, Zone tab](#).

Owned Line Records Display

The Owned Line Records Display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Line and Transformer Display](#). In addition to the information available for viewing from the [Line and Transformer Display](#), the Owned Line Records Display also shows the owner number and percentage of ownership. Currently, Simulator allows up to four owners for one generator, with a total percentage ownership between 0 and 100%. To make changes to the owners and ownership percentages for a transmission line, right-click on the grid and choose Show Dialog and select the [Owners tab](#).

Jacobian Display

The Jacobian display is a matrix showing the system Jacobian matrix for the currently loaded Simulator case. This display can be very useful for educational purposes. Keep in mind that for a large case, this display can contain a very large matrix. It is possible to right-click on this display and save the grid to a Matlab formatted file, or to export the grid to an Excel spreadsheet. Be aware that a Jacobian matrix from a large will often exceed the size limitations of an Excel spreadsheet.

Ybus Display

The Ybus display (bus admittance matrix) is a matrix showing the system Ybus for the currently loaded Simulator case. This display can be very useful for educational purposes. Keep in mind that for a large case, this display can contain a very large matrix. It is possible to right-click on this display and save the grid to a Matlab formatted file, or to export the grid to an Excel spreadsheet. However, Excel does have limitations on the number of rows and columns that could quickly be exceeded with a Ybus from a large case.

Chapter 9: Solving and Simulating a Case

This chapter describes the controls and general tools for solving a load flow in Simulator. These topics are used to customize and control the time domain and single Power Flow Solutions in the Run Mode. Additional topics also describe ways to view results and data graphically using auto-generated graphical displays.

The following material is included:

- Solution Options
- Solution and Control
- Transactions
- Charts
- Bus View Online
- Substation View Online

Simulator Options

Simulator provides a flexible environment for simulating power system operation by offering you access to a number of customizable options. The **Simulator Options Dialog** houses pages of options that you can customize to tailor the program to your needs.

To display the Simulator Options Dialog go to the [Options](#) ribbon tab, and choose **Simulator Options** from the Case Options Ribbon Group. The Simulator Options button is also available on the [Tools](#) ribbon tab on the [Power Flow Tools](#) ribbon group.

If a timed simulation is active when you open the dialog, it is automatically paused and will not resume until you close the dialog.

There are seven categories of options for the Simulator Options dialog:

[Case Information Display Options](#)

[Environment Options](#)

[File Management Options](#)

[Limits Options](#)

[Message Log Options](#)

[Online Options](#)

[Power Flow Solution Options](#)

Power Flow Solution Options

The Power Flow Solution Tab offers various options regarding how Simulator solves the power flow problem. There are six sub-categories on the Power Flow Solutions tab: [Common Options](#), [Advanced Options](#), [Island-Based AGC](#), [DC Options](#), [General](#), and [Storage](#). Each of these categories is shown on tabs. Many of the options in the Power Flow Solution options will be of interest only to advanced users of the package.

Power Flow Solution: Common Options

The following options are all contained on the Common Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Power Flow (Inner) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

MVA Convergence Tolerance

The MVA convergence tolerance serves as a measure for determining when the inner power flow loop of the Power Flow Solution process has reached an acceptable solution. The MVA mismatch is computed as the maximum real or reactive mismatch at any bus in the system. Usually, this value should be around 0.1 MVA. If you are having difficulty solving a particular case, it may be helpful to temporarily increase the MVA Convergence Tolerance to drive the solution closer to the actual solution, and then re-solve from this solution using the smaller MVA tolerance.

Maximum Number of Iterations

This option defines the maximum number of iterations Simulator will perform during the Power Flow Solution process in an effort to converge to a solution. If Simulator must exceed this number of iterations, it assumes that the power flow case is not converging and will terminate the solution process. If Simulator is configured to represent non-converging power flow cases as blackouts, the screen will turn gray and the blackout warning message will appear.

Do Only One Iteration

If checked, then Simulator will only perform one iteration of the load flow solution process when **Single Solution** is clicked, regardless of the Maximum Number of Iterations setting. This is useful when playing with a case with which you are experiencing difficulty.

MW Control (Outer) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Automatic Generation Control (AGC)

If checked, the enforcement of the Generation Re-dispatch to account for MW interchange constraints for all areas is disabled. By default, this option is not checked.

Enforce Generator MW Limits

If checked, then generator minimum and maximum MW limits are enforced for all generators whose *Enforce MW Limits* field is set to true. See [Generator Information Dialog](#) for more information. Otherwise, generator MW limits are not enforced. Note that when using the economic dispatch or the Optimal Power Flow, the generator MW limits are always enforced regardless of user-settings.

Topology Processing Options (These options are only available with the [Topology Processing](#) add-on)

Use Topology Processing

This option is used for all Simulator applications: power flow, contingency analysis, sensitivity calculations, etc. This option should ALWAYS be checked for real-time cases, while it should be unchecked for planning cases. If a full-topology model was read and this option is not checked, the power flow will try to solve the model without removing the switching devices resulting in a very large [ill-conditioned Jacobian](#). This option is also found on the [Topology Processing Dialog](#).

Controller (Middle) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Checking Gen VAR Limits

If checked, the Mvar limits are ignored for all the generators in the case during a Power Flow Solution. By default, this option is not selected.

Check Immediately

Before entering the inner power flow loop to solve the power flow equations, a decision is made about each generator in the case as to whether it will be treated as a PV or a PQ bus. It is assumed that a PV bus will maintain its setpoint voltage, while a PQ bus will maintain a constant reactive power output (for instance when the generator is at a Mvar min or max limit). Normally, the power flow equations are completely solved in an inner power flow loop using this assumption. After completing the solution, generator Mvar limits are checked to see if they have been reached (must change from PV to PQ), or if at a limit if the limit may be backed off of (must change from PQ to PV). If any generators change PQ/PV status, then the inner power flow loop is resolved.

When the **Check Immediately** option is checked, the generator Mvar limits are checked after *every* inner power flow loop iteration instead to see if generator limits have been reached, or if a generator limit may be backed off. In some situations performing this check inside the inner power flow loop can help with convergence, however performing this check does slow down the solution process. By default, this option is not selected.

Disable Switched Shunt (SS) Control

If checked, automatic control of switched shunts is disabled in all areas. By default, this option is not selected.

Disable LTC Transformer Control

If checked, automatic control of LTC transformers is disabled in all areas. By default, this option is not checked.

Disable Phase Shifter Transformer Control

If checked, then automatic control of phase shifting transformers is disabled in all areas. By default, this option is not checked.

Transformer Stepping Methodology

Choose either *Coordinated Sensitivities* or *Self-Sensitivity Only*. The default value is *Coordinated Sensitivities*.

When the regulated value of tap-changing or phase-shifting transformers move outside of their regulation range, then Simulator attempts to bring those transformers back inside their range. By default Simulator determines all the tap-changing and phase-shifter transformers which are out-of-range and coordinates the movement of these transformers in an attempt to bring all the regulated values back into range. This is what is called *Coordinated Sensitivities*. Generally this results in a better convergence, however it also can be slower when a large number of transformers are out-of-range together. Because it gets slower with a large number of transforms, if more than 50 transformers are involved in the switching, Simulator will always use the *Self-Sensitivity Only* option. As the solution process continues however, the number of transformers moving will reduce and eventually the *Coordinated Sensitivities* will be used again.

Choosing *Self-Sensitivity Only* will modify this methodology so that each transformer only looks at the sensitivity of its regulated value with respect to changing its own tap or phase. This calculation is faster, but may result in convergence problems due to transformers that interact with one another.

Prevent Controller Oscillations

Sometimes, a power flow will fail to converge because certain automatic controls such as Mvar limit enforcement at generators, transformer tap switching, and shunt switching oscillate between their control bounds. These oscillations very often are due to modeling inaccuracies. If this option is checked, Simulator will automatically detect such oscillating controls and fix them at their current value so that they no longer oscillate. You may find this option helpful if you feel that the modeling of automatic controls in your system is inaccurate.

Maximum Number Controller Loop Iterations

As part of the solution process, the outer loop of the solution algorithm is a check of any necessary controller changes due to changes in controlled values from the last iteration of the Newton-Raphson load flow solution. The maximum number of loops through the control change algorithm can be set here. This is not the same as the Maximum Number of Iterations, which applies to the actual Newton-Raphson inner loop algorithm, which solves that actual power flow.

Power Flow Solution: Advanced Options

The following options are all contained on the Advanced Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Dynamically add/remove slack buses as topology is changed

If checked, Simulator dynamically chooses a new slack bus for the islands if the user-input does not adequately define slack buses. If a slack bus can not be determined for an island, the island cannot be solved and will be isolated and ignored during the load flow solution. Also if there are multiple slack buses specified in the same island, Simulator will choose only one of them as the slack bus.

When allowing dynamic slack bus assignment, the choice of the island slack bus is managed in Simulator by internally maintaining a list of buses the user has identified as slack buses while in Edit Mode. In Run Mode, electrical islands are then dynamically determined from the system topology. As branch statuses are changed, this list of islands (and slack buses) will automatically change. An island will be considered viable only if it meets the following criteria.

1. At least one island bus has some on-line generation with a positive Maximum MW Limit and is NOT set to regulate the voltage at a bus in another island.
2. At least one island bus has a closed load (Note: It could still be a 0.0 MW load) OR the island is connected to another island by a DC transmission line.

After the viability of each island is determined, a slack bus will be chosen for each viable island. (Note: appropriate messages are written to the log when buses are turned into slack buses or removed as slack buses). If an island has exactly one user-identified slack bus, then that is used as the slack. If an island has greater than one user-identified slack bus, then the user-identified slack bus which has the largest maximum generator MW output is used as the slack bus. If an island has no user-identified slack buses, then one of the buses in the island will be automatically chosen as the slack bus. When choosing this bus, the four criteria listed below are used. If buses are found that meets all four criteria, then of those buses the one with the largest maximum generator MW limit will be used. If no bus is found which meets these criteria, then of the buses that meet the first three criteria, the one with the largest maximum generator MW limit will be used. Again, if no bus is found, the first two criteria are used and so on.

1. At least one generator at the bus is set to AVR = YES
2. The sum of the Maximum MW limits for generators at the bus is less than 5,000 MW (if there is more, these are probably a fictitious MW limits)
3. Generators at the bus regulate their terminal bus (this will be required by the slack bus anyway)
4. There is only one generator at the bus (it is just easier for the user to keep track of then)

Post Power Flow Solution Actions

Clicking this option will open the [Post Power Flow Solution Actions dialog](#), where the user can specify a list of actions to be executed at the end of every Full AC power flow solution.

Power Flow (Inner) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Power Flow Optimal Multiplier

If checked, the Newton solution process will not use the optimal multiplier. The optimal multiplier is a mathematically calculated step size for the Newton's Method iteration that prevents the mismatch equations from increasing between iterations. A small value indicates that moving in the direction of the Newton step will increase the mismatch equations. When the optimal multiplier becomes too small, this is an indication that the power flow is not going to solve successfully and Simulator will stop instead of creating extremely large mismatches. If this happens at a point where the solution is not within the allowed tolerance for the Newton process, the Newton process will result in a failed convergence to a valid solution.

Initialize From Flat Start Values

When checked, each Power Flow Solution is started assuming that all voltage magnitudes and generator set point voltages are unity and all angles are zero. By default, this option is not selected. Some power flow problems can be very difficult to solve from flat start assumptions. Therefore, use this option sparingly.

Minimum Per Unit Voltage for Constant Current Loads

This option is used to model the impact that falling voltage has on loads throughout the system. The default value is 0.5.

For constant current loads with a terminal bus below this per unit voltage, the value of the load will fall off using a sine function so that the load is zero at zero terminal voltage, and so that the derivative is continuous at the minimum per unit voltage for constant current loads.

Minimum Per Unit Voltage for Constant Power Loads

This option is used to model the impact that falling voltage has on loads throughout the system. The default value is 0.7.

For constant power loads with a terminal bus below this per unit voltage, the value of the load will fall off using a cosine function so that the load is zero at zero terminal voltage, and so that the derivative is continuous at the minimum per unit voltage for constant current loads.

Control (Middle) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Treating Continuous Switched Shunts (SSs) as PV Buses

Continuous switched shunts are normally treated the same as a generator bus inside the inner power flow loop (they are treated as buses with fixed power and voltage). Checking this option will cause the continuous switched shunt to be treated as a constant impedance in the inner power flow loop with all switch occurring in conjunction with other switched shunts in the voltage control loop instead.

Disable Balancing of Parallel LTC Taps

Simulator has the capability to attempt to balance tap positions of parallel transformers, in an attempt to avoid parallel transformers from going to opposite tap settings, inducing loop flow through the parallel transformers. Checking this option disables the automatic balancing of parallel transformers. The only transformers that will be balanced are those in parallel between the same terminal buses, or those in parallel between terminal buses that are connected with very-low impedance branches.

Model Phase Shifters as Discrete Controls

If checked, then phase shifters will switch tap positions discretely based on the tap step size of the phase shifting transformer. By default, this option is not checked, which means the phase shifters will switch continuously, independent of the tap step size.

Minimum Sensitivity for LTC Control

This option specifies the minimum-voltage-to-tap sensitivity for LTC transformers. All transformers having an absolute value of voltage-to-tap sensitivity below this value are automatically disabled from automatic control. This prevents Simulator from changing transformer taps that have little effect on their controlling voltage. The [Transformer AVR Dialog](#) shows the voltage-to-tap sensitivity for each voltage-controlling transformer.

Disable Angle Rotation Processing

At the end of a power flow solution, by Simulator looks at the bus voltage angles in the case to see if any are near +/- 180 degrees. If the angles fall outside of +/- 160 degrees, then all angles in the island will be rotated by the same amount so that the angle range in the islands is equally spaced around zero degrees. Check this option to disable this feature.

Sharing of generator Vars across groups of buses during remote regulation

When several buses have generators that control the voltage at single bus, this option determines the method used to determine each buses "share" of the MVar support. There are three options

1. Allocate across buses using the user-specified remote regulation percentages. This option is what is used by default and most closely matches the sharing seen in RAW files.
2. Allocate so all generators are at same relative point in their [min .. max] var range. This option most closely matches the sharing seen in a few EMS solutions PowerWorld has seen.
3. Allocate across buses using the SUM OF user-specified remote regulation percentages. This option most closely matches the sharing seen in EPC files.

It should be noted, generators at the same bus always allocate vars so they are at the same relative point in their [min..max] MVar range. How vars are allocated to generators at the same bus has no effect outside of that bus so options are not given for this sharing.

Include Loss Penalty Factors in ED

If checked, the economic dispatch calculation will consider losses in determining the most economic generation dispatch. Otherwise, the generation dispatch calculation will disregard system losses.

Enforce Convex Cost Curves in ED

The economic dispatch algorithm attempts to set the output of all generators that are set to be automatically controlled so that the system's load, losses, and interchange are met as economically as possible. The algorithm is guaranteed to reach a unique solution only when all generator cost curves, which model the variation of the cost of operating a unit with its output, are convex. If this option is checked, Simulator will identify units whose operating point is outside the convex portion of the cost curve and set them off automatic control.

Post Power Flow Solution Actions Dialog

The Post Power Flow Solution Actions dialog describes a list of actions that are executed at the end of every Full AC power flow solution, which means they are not performed for DC solutions. Normally, these actions would all have Model criteria specified.

To open this dialog, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. Next, go to the **Power Flow Solution** category. Then click on the **Advanced Options**. Finally click the button labeled **Define Post Power Flow Solution Actions**.

Check the **Do Not Used Post Power Flow Solution Action List** check-box to disable using any actions defined in the list.

The action list display identifies the actions that comprise the post power flow solution action list. Actions can be inserted or deleted by using the local menu on the dialog. Right-click on the display and select **Insert** or **Delete**. Actions are inserted via the [Contingency Element Dialog](#).

The action display is a type of [Case Information Display](#) and thus shares many common to all other case information displays.

The Action List Display always contains the following fields:

Actions

This shows a string which describes the action. You may customize the format of the string that describes the actions by right-clicking on the Action List Display and choosing **Display Descriptions By**, and then choosing either Name, Num, Name/Num, PW File Format by Numbers, PW File Format by Name/kV or PTI File Format.

Model Criteria

Simulator allows you to define Model Criteria, which consist of both [Model Conditions](#) and [Model Filters](#). These specify a criteria under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the Model Condition and Filter option on the [Contingency Options Tab](#). You can open a dialog to define Model Filters or Conditions by right-clicking on the Action List Display and choosing **Define Model Criteria**.

Status

Three options are available for this field

- CHECK : The action will be executed only if the Model Criteria is true. It will also be executed if no model criteria is specified. This is the default setting
- ALWAYS: The action will always be executed, regardless of the Model Criteria.
- NEVER: The action will never be executed, regardless of the Model Criteria.
- POSTCHECK: This action is checked AFTER the other Check and Always actions have been performed and the load flow solution solved. If the criteria specified for the Postcheck action are met in the resulting load flow solution, then this action is taken and the load flow is again resolved. This will recursively occur for all Postcheck actions until either all postcheck actions have been taken, or the criteria for all remaining postcheck actions have not been met.

Note that the Never action allows you to disable a particular action without deleting it.

Comment

A user-specified comment string that can be associated with this action. While this comment is not used by Simulator in any way, these comments can be saved to be loaded at a later time. This is provided for the user to add comments regarding the action. For example, for an action with a Model Criteria you may could add a sentence explaining why the action is only performed under the specified criteria.

Power Flow Solution: Island-Based AGC

The following options are all contained on the Island-Based AGC tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

The Island Based AGC options allow the user to choose to dispatch generation by island instead of by area or super area. The options available for dispatch here are:

Disable (Use the Area and Super Area Dispatch Settings)

When this option is selected, Island Based AGC is being used, and generators are dispatched according to the usual area or super area generation dispatch.

Use Participation Factors of individual generators

When selected, the island-based AGC is used, based on the individual participation factors of each generator within the island. Area and super area ACE requirements will be ignored, and all generators in the island will be dispatched to serve load, losses, and any DC transfers to other islands.

Note that if all generators with AGC = YES have participation factors also equal to 0.0, then all make-up power will come from the island slack bus.

Calculate Participation Factors from Area Make Up Power Values

The AGC will dispatch generation by use area make up power values, instead of by individual participation factor. What this means is that each area will be assigned a make up power value, similar to assigning a participation factor to a generator. Based on the make up power value of each area in the island, the amount of the generation dispatch needed will be divided amongst each area based on its value. Higher value will account for more of the generation dispatch than areas with smaller values. The within each area, the generation dispatch is handled on an individual generator participation factor basis, where each generator will account for a portion of the dispatch that was assigned to its area.

To specify the make up power values for each area, click on the **Specify Are Make Up Power Values** button. Note that these settings are the same as those used by [Contingency Analysis Options Make Up Power](#), therefore changing these values will affect the contingency analysis.

For example, three areas have participation factors of 2, 1 and 1, respectively. If the total generation redispatch in the island is 100 MW, then area 1 will account for $2 / (2+1+1)$, or 50%, of the total. Therefore area 1 is expected to redispatch by 50 MW. If area 1 then has two generators with participation factors of 4 and 1, they will account for the 50 MW by picking up $4 / (4+1)$ and $1 / (4+1)$, or 80% and 20%, respectively, of the 50 MW needed from the area.

Note that if all areas have **CTG Make Up Gen** equal to 0.0, then all make-up power will come from the island slack bus.

Dispatch using an Injection Group (Loads and Generators will respond)

Checking this option will allow for the island dispatch to be covered by change in generation and/or load defined in an [injection group](#).

Additional options are available when dispatching based on an injection group:

Allow only AGC Units to Vary

If this option is checked, only units whose AGC status is turned on will be allowed to participate in the injection group dispatch.

Enforce unit MW limits

If checked, then each generator's defined MW limits will be strictly adhered to during the redispatch.

Do not allow negative loads

When checked, loads included in the injection group are not allowed to drop below zero MW or MVAR load demand.

How should reactive power load change as real power load is ramped?

You can choose to keep the ratio of real and reactive power constant for each load that is included in the injection group, or you can specify a constant power factor that the MVAR value will be determined from when the MW value is changed.

Island AGC Tolerance

This sets the AGC tolerance in MW for the real power balance of each island for all options except **Disable (Use the Area and Super Area Dispatch settings)**. Generation must equal load plus losses plus DC line exports in each island. If necessary, each island slack bus output will vary during the [Power Flow \(Inner\) Loop](#) solution to maintain the real power balance for the island. If it varies more than the specified tolerance and if a form of Island-based AGC is specified, then adjustments to AGCable units will be made in the [MW Control \(Outer\) Loop](#) according to the selected Island-based AGC option.

Power Flow Solution: DC Options

The following options are all contained on the DC Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Use DC Approximation in Power Flow

When this box is checked, Simulator will solve the load flow using the DC Approximation method. When not checked, Simulator performs the modified Full Newton AC load flow algorithm. Note that once a case, especially a large one, has been solved using a DC approximation, it tends to be quite difficult to revert back to the Full Newton AC load flow from a solved DC approximation.

DC Power Flow Model

Note: These choices also affect the calculation of sensitivities on the [PTDF](#), [LODF](#), and [TLR](#) dialogs and [ATC Tool](#) calculations.

Generally, the *only* inputs that affects the DC Power Flow equations are line impedances and line status (open/closed). However there are three possible modeling choices (for which there is not general agreement in the industry) that will affect the "DC equations" in small ways. These choices are made here on the DC Power Flow Model options

Ignore Series Resistance (r) or Ignore Series Conductance (g)

The series term of a transmission line consists of an r and x value which represent a complex number impedance. In the DC power flow approximation, the value that used is the imaginary part of the inverse of impedance (called admittance)

$$g + jb = 1 / (r + jx) = r / (r^2 + x^2) - j x / (r^2 + x^2)$$

Thus

$$g = r / (r^2 + x^2)$$

$$b = - x / (r^2 + x^2) \quad *** \text{ This term is used in the DC power flow equations}$$

The only term that is used in the DC power flow equations is the b term. Some say that a DC power flow means that r = 0 which means that b = -1/x and g = 0. Others say that a DC power flow only means assuming that g = 0, which means that b = - x / (r^2 + x^2). There is not a good consensus in the industry as to what is correct regarding this, so Simulator offers an option as to whether you ignore r or ignore g. The default option in PowerWorld is to ignore the resistance.

Ignore Transformer Impedance Correction Tables and Ignore Phase Shift Angle Effects

These are both ignored by default.

When a transformer impedance correction table is specified for a transformer, then the series impedance of the transformer will vary by a multiplier with the tap or phase of the transformer. The tap ratio is not generally relevant to a DC power flow, however the phase-shifter angle could vary when using the DC approximation. This option specifies whether to ignore the impact of this impedance correction. For phase shifters the multiplier is normally between 1.00 and 1.50 and thus tends to *increase* the impedances as you move away from zero degrees.

As a branches phase shift varies, the DC power flow equations could be modified to take this into account. This effect results in the DC power flow equations for which the "b" term above is multiplied by the cosine of the phase angle across the line. This effect tends to *decrease* the impedance as you move away from zero degrees.

Because the impedance correction tends to increase impedance and the phase effect tends to decrease impedance it is reasonable to ignore both of these, and this is done by default. The other issue with not ignoring these is that both create a situation where the equations for the DC

approximation are dependent on the system state. In other words, the matrix equations become a function of the phase-shift angle. This results in a situation in which the matrices must be recalculated and re-inverted each time a phase angle is moved. This removes some of the advantages of the DC approximation. This can be especially problematic when using the OPF or SCOPF.

Compensate for Losses by Adjusting the Load

Traditionally a DC load flow is treated as lossless. However, you can approximate the loss in the load flow by artificially adjusting the load in the case to include estimated losses. To do so, click on the **DC Loss Setup** button to open the [DC Power Flow Loss Setup](#) dialog for setting the DC Loss Multipliers.

Compensate for Reactive Power Flows by Adjusting the Branch Limits ...

One issue with DC load flow is how to treat the inclusion of reactive power flows. In a standard DC load flow, the reactive flows are typically ignored, and the MW flow of a branch is compared to its original MVA branch limit. However, Simulator also gives you the option to instead solve the DC load flow and compute approximate MVAR flows by assuming the voltages in the system remain constant when the DC load flow is solved.

Compensate for Dispatch Sensitivities with User-Specified Values

This option allows for the bus MW loss sensitivities to be used in the OPF and ED dispatch algorithms, if the type of loss sensitivity on the [General](#) tab is set to User-Specified.

DC Power Flow Loss Setup

To open the DC Power Flow Loss Setup dialog, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. On the Power Flow Solution page, click on the tab labeled **DC Options**. Next, click the button labeled **DC Loss Setup**. This button will only be enabled when you check the box to **Use DC Approximation in Power Flow/OPF/SCOPF**.

The DC Power Flow Loss Setup dialog gives you a location to apply approximate losses during a DC load flow solution. The losses can be approximated by scaling the loads in the case to include an approximation of losses. Loss multiplication factors can be applied individually by bus, or as a group by area or by zone. Note that loss multipliers by area or by zone are just quick ways for setting the bus multiplication factors for all buses in the group. You will see the value reflected for all buses in the Buses page. If you wish to apply the same multiplication factor to the entire case, you can simply set the Case DC Loss Multiplier at the bottom of the dialog. This will automatically set all buses in the case to have the same DC Loss Multiplier specified.

Once the DC Loss Multipliers have been set, click OK to save the multipliers.

Power Flow Solution: General

The following options are all contained on the General tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Assumed MVA Per-Unit Base

This option allows the user to specify the MVA base to be used for the entire case. By default, this value is set to 100 MVA. Click on the **Change System Base** button to change this value and then update all the internal structures to store the proper values on the new base.

Bus Loss Sensitivity Function

Bus loss sensitivities indicate how island or area losses change with power injection at the bus. Here you may choose to forego the calculation of bus loss sensitivities or to base them on island losses or area losses. If the case consists of only one island, which, by definition, corresponds to the entire system, then the bus loss sensitivities are measured with respect to total system losses. If the bus sensitivities are set to User-Specified, the sensitivities will remain at their last calculated values, according to the loss function type previously specified when the [loss sensitivities](#) were calculated.

Monitor/Enforce Contingent Interface Elements

This global location allows you to determine how contingency elements in an interface should be treated in Simulator. You can choose to never include the impact of contingent elements on interface flow, to only include contingent element impacts in the standard power flow or optimal power flow routines, or in all solution routines including contingency analysis and security constrained OPF.

It is not uncommon to ignore the impact of contingent elements when using the contingency analysis or security constrained OPF tools, as they are already processing lists of contingencies and evaluating flows on interfaces. Ignoring contingent elements within interface definitions allows for a determination of the impact of other contingencies on the flows of the non-contingent elements forming the interface, without impact from additional contingent element considerations.

Power Units for Display

Allows switching between displaying power values in Mega- units or Kilo- units.

Power Flow Solution: Storage

The following options are all contained on the Storage tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Simulator offers the ability to restore either the last power flow solution state or the state of the system immediately before the last solution attempt. If your system has insufficient memory and you are working with large systems, you may wish to disable one or both of these options.

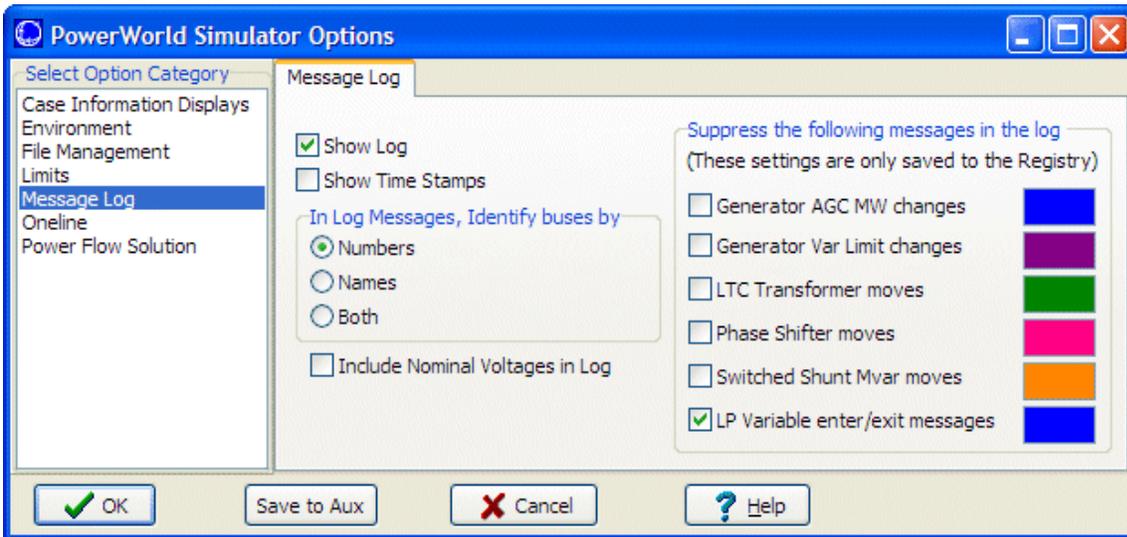
Disable "Restore last solution"

Restoring the last solution will undo any changes made to the data that were made after the last successful solution and return the case to the last valid solution. For large cases, the amount of memory required to store the last solution can be significant. If this option is checked, Simulator will not store this information in memory, and the last solution cannot be restored if a solution fails.

Disable "Restore state before failed solution attempt"

Restoring the state before failed solution attempt will undo only the attempted solution, but will retain any changes to data that were made before the solution process. This allows the user to return to the point just before the solution in order to add or remove changes in an effort to obtain a valid solution. For large cases, the amount of memory required to store the state before a solution attempt can be significant. If checked, Simulator will not store the state information in memory, and the state before the failed solution cannot be restored.

Message Log Options

**Show Log**

If checked, the message log is displayed. The message log shows detailed results of each Power Flow Solution. Usually this log is NOT shown. However, if you are running into problems with a simulation case, it can prove useful for debugging the case.

Show Time Stamps

Checking this option will display the date and time at which each message is posted in the message log.

In log Messages, Identify buses by

This option allows the user to specify how to identify the buses in log messages. The options are by numbers, by names, or by both numbers and names.

Include Nominal Voltages in Log

If checked, this option will make the buses to be displayed with their nominal voltages after their name.

Suppress the following messages in the log

Checking the check-boxes in this option will remove the corresponding message writing to the log, thus speeding up the computation process. The boxes to the right of the messages indicate the color with which the messages will be displayed in the log.

An example of how these colors are reflected in the message log is shown in the following image.

Environment Options

The Environment page provides you control over a number of display and simulation options. The first section of the page contains a list of check boxes that you can use to designate the content of the oneline displays.

These options include:

Do Not Solve While Animating (Display Only)

If checked, Simulator only displays the case; it does not solve the power flow equations. System flows are determined by the initial values in the case file. This option should be checked if you simply want to use Simulator to visualize a case that has already been solved. The advantage of the display-only mode is that animation is significantly faster, particularly for large cases. The drawback to the display-only mode is that the power flow equations are not automatically solved at each time step; you must explicitly call for a Power Flow Solution using either the **Single Solution** button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab or choose one of the solution options under the **Solve** menu in the [Power Flow Tools](#) ribbon group.

Play/Animation Solution Method

This option is available only for OPF releases of Simulator. Use it to indicate whether Simulator's repetitive power flow should perform a normal power flow solution or an optimal power flow (OPF) solution.

Solution Animation

Check Auto Start Solution Animation to automatically start the animation after loading the case.

Check Auto Solve On Load to automatically solve the power flow solution when the case is loaded.

Show Log

If checked, the message log is displayed. The message log shows detailed results of each Power Flow Solution. Usually this log is NOT shown. However, if you are running into problems with a simulation case, it can prove useful for debugging the case.

Disable Showing Blackouts

You can dramatize a power flow case's failure to converge by representing it as a blackout. The background of the oneline diagram will become a dark shade of gray, and a message box will appear to announce that the system has experienced a blackout. To disable this behavior, select the Disable Showing Blackouts box. Representing the failure to converge as a blackout can be very helpful for presentation purposes. Very often, the power flow's failure to converge can be traced to the system's inability to serve the load demand, a situation that requires that load be "blackened out," or shed, to restore the system to a viable operating state. Thus, displaying the failure to converge as a blackout has physical significance.

Disable AGC When Manually Changing Generator MW

When this option is checked, changing the MW output of a generator manually will automatically remove a generator from [Automatic Generation Control](#). If you wish for generators to maintain their automatic generation control settings following a manual change of MW output, you must uncheck this option.

Open Associated Oneline Diagrams

By default, Simulator has always attempted to open any diagrams that were open with a case file when the case file was last closed. However, now you may uncheck this option and Simulator will not attempt to open any oneline diagrams when a case is loaded.

Auto Open Bus Records if No Online

If a case is opened in Simulator which does not have an associated online diagram, then Simulator will automatically open the [Bus Records](#) case information display if this option is checked.

Allow to Prompt Dialog to Create Online Diagram

If a case is opened in Simulator which does not have an associated online diagram, then Simulator will prompt a message dialog that allows the user to create a blank online diagram.

Auto Set Case as Base Case for Difference Flows when Loaded

This option allows the case to be set as a base case automatically after being loaded. This can be done always or never, or only if the base case has not been set before opening the case.

Reset Default Positions and Styles for all Dialogs

Click this button to reset the default position, size and Free-Floating/Contained style of the dialogs. See [Window Basics](#) for more information.

Custom Colors

This option allows the user to edit the custom colors used throughout all Simulator color dialogs.

Clock Style

The clock serves as a timer for timed simulations by showing the current time, the start time, and the end time of the simulation. You can choose to hide the simulation clock by specifying a clock style of *None*. Otherwise, to display the clock in its own window, choose *Dialog*, and to display the clock on the program's status bar, select *Status Bar*.

Measurement System

This option allows the user to choose English (Imperial) or Metric (SI) units for system measurements. By default, this option is set to English units.

Recently Used File List Entries

The maximum number of file names and locations stored in the History List of the File menu.

Undo Memory Limit Per Online

The approximate maximum number of Megabytes allowed to be used in storing actions carried out when editing a online diagram, which can be undone at any time.

Online Options

These options are available on the Online tab of the [PowerWorld Simulator Options](#) dialog. It is important to note that the options on this dialog apply to *all* online diagrams. There are also a large number of options which apply only to an individual online diagram. The diagram-specific options are specified on the [Online Display Options Dialog](#).

Visualizing out-of-service elements

These three options allow you to choose how objects on the diagram should appear when they are representing a power system device that is currently "open" or "out-of-service." The three options are Blink, Use dashed lines, and Draw and X through off-line generators. The first two options apply to any online object, while the third option is specific to generator objects only.

Main Online File

This option is used to identify the primary online diagram to use with the case. The main online is the file that is displayed when you first open the case. The dropdown box lists all the online files that reside in the same directory as the case. Select one of these files, or enter the full path of the online you want to use if it does not appear in the dropdown box.

Use Default Online File

You can command Simulator to open a particular online diagram file if it cannot find a online diagram file for the case you are trying to open. For example, there is no online diagram associated with a PSS/E raw data file when you first read it into Simulator. However, if your application is such that you will always use the same online file whenever you open a PSS/E raw file, check the **Use Default Online File** to have Simulator open the online identified in the **Default Online File** box whenever it encounters a case that has no associated online. The default online must exist in the same directory as the case you are trying to open.

Show Online Hints

If checked, pop-up hints will appear when you drag the mouse over an object in the online. These give information about the object; for example, for a generator the pop-up hint displays the bus number, generator ID, the MW output and the MVAR output. You may also customize the pop-up hints for each kind of display object by choosing [Options > Custom Hint Values](#) from the menu.

Show X,Y Coordinates

If checked, the (x,y) location of the cursor is monitored in the status bar at the bottom of the screen. The (x,y) location of the cursor is only shown in Edit Mode. By default, this option is selected.

Save Contour Image with Online File

If checked, and if a contour is being displayed on the online diagram, Simulator will store the contour with the online diagram when you save the case or save the online. It will store the contour as a bitmap.

Display Unlinked Elements in Run Mode

Typically, unlinked graphical objects are not visible in Run Mode. Checking this option will allow the display of unlinked elements during Run Mode.

Enable Mouse Wheel Zooming

This option allow the use of the mouse wheel to zoom in and out in the currently selected online diagram.

Minimum Screen Font Size

The minimum font size at which text is visible on the screen. When text is rendered to the screen, any text which is smaller than this font size will not be rendered. This is useful when zooming out on a oneline diagram where a lot of text might become cluttered or hard to read on the screen.

Minimum Print/Copy Font Size

The minimum font size at which text can be printed or copied. This is useful if the application or printer you are sending to can or cannot display smaller fonts. When text is printed, any text which is smaller than this font size will not be printed. Normally the Minimum Print/Copy Font Size is smaller than the Minimum Screen Font Size because the resolution of the printed document is much higher than the screen resolution.

Transformer Symbol

Since transformer representation varies in different countries, this option allows the user to represent transformers as coils or circles. By default, transformers are represented as coils.

Save Onelines when Saving Case

By default, Simulator always saves any oneline diagrams that are open when the user chooses to save the case (pwb) file. This option allows you to choose to be prompted to save oneline diagrams when a case file is saved, or to never save onelines when the case file is saved.

Edit Oneline Browsing Path

This option applies when you have [Oneline Links](#) included on a oneline diagram. Rather than specify the full path and name of a oneline diagram as a oneline link, you can specify the file name only. When the link is clicked in Run Mode, Simulator will check all directories listed here, in order, to try and find the oneline file name stored with the link. This browsing path is also used when looking up [Shapefile Database R cords](#) and when finding the substation layout, URL, or command specified on the [Substation Information Dialog](#).

Line Navigation Arrows

These settings control whether the Pixel Size of the arrows, how long in seconds it should take to pan to the line end locations, and whether to show navigation hints as your mouse hovers over the arrow. See [Oneline Zoom and Panning](#) for more information.

Case Information Display Options

PowerWorld uses numerous [case information displays](#) to show `javascript:RelatedTopic1.Click()` in tabular format. The options presented on this page of the PowerWorld Simulator Options Dialog control some of the general features of the case information displays.

Enterable Field Color

Fields whose values can be directly entered on the case information displays are colored navy blue by default. Click on the color field to change the color of enterable fields, or click the **Change** button.

Toggleable Field Color

Fields whose values can be toggled (changed) by left-clicking on them are colored green by default. Click on the color field to change the color of toggleable fields, or click the **Change** button.

At or Exceeding Limit Color

Fields whose values are at or exceeding a limit, are colored red by default. Click on the color field to change the color of such fields, or click the **Change** button.

Normal Field

Fields that cannot be modified directly from the case information display are colored black by default. Click on the color field to change the color of such fields, or click the **Change** button.

Special External Field

Fields that can be modified through an auxiliary file or by pasting data into a sheet from a spreadsheet, but cannot be changed in a case information display, are colored black by default. Click on the color field to change the color of such fields, or click the **Change** button.

Field not presently used

Fields whose values are overridden by other instances of the program are colored gray by default. Click on the color field to change the color of such fields, or click the **Change** button.

Background

Background of cells is colored white by default. Click on the color field to change the color of the background, or click the **Change** button.

Heading Background

Background of column and row headings is colored light gray by default. Click on the color field to change the color of the heading background, or click the **Change** button.

Data Fill Background Color

Background of selected cells when propagating values is colored yellow by default. Click on the color field to change the color of the heading background, or click the **Change** button.

Set Case Info Factory Default Colors

Clicking the **Dark Colors** button will reset the field and background colors to the defaults mentioned above, which comprise dark colors for fields, and light colors for backgrounds. Clicking the **Light Colors** button will set field and background colors to a specified set of light colors for fields and dark colors for backgrounds.

Highlight Objects with Selected? = YES

Check this option to automatically highlight all case information rows that represent a model object that has its "Selected" field set to YES. The row will then be highlighted using the color specified on this dialog. This option will apply to all case information displays representing all types of objects.

Save as Auxiliary File Data Format

This option allows the user to decide if the auxiliary files that can be saved from case information displays should be space delimited files or comma delimited files.

Key Fields to Use in Subdata Sections

This option allows the user to decide whether to use primary or secondary key fields or labels for the objects saved in the subdata section of an auxiliary file. In addition to being applied to the subdata section of auxiliary files, this option is also used in some special cases to dictate how elements are identified. The [Labels](#) topic contains additional information about this.

View/Modify Default Font

Clicking this button brings up a font dialog from which you can choose the font in which case information displays should show their data. Selecting a new font, font size, style, or color and pressing OK will change the default font, so that all case information displays will then employ a font having the selected properties. Not an individual case information display can override this setting by choosing a custom row height or font on the [Display Options tab of the Display/Column Options dialog](#).

Default Row Height

This option sets the default height of the rows in the case information displays. This field may need to be changed depending on the screen size and font size of the computer. By default, the height is set to 13. Not an individual case information display can override this setting by choosing a custom row height or font on the [Display Options tab of the Display/Column Options dialog](#).

Show Grid Lines

Grid lines are shown by default in case information displays. Unchecking this box will remove the grid lines from the displays.

Column Headings

This option allows you to choose whether the column headings of the case information displays are the normal column headings or are the variable names of the type of data stored in each column. Variable names are the strings used when writing data out to an [Auxiliary File](#). Change the headings to variable names can be helpful when looking at or creating auxiliary files.

Use Word Wrap

Check this box to wrap the heading column text, breaking it in several rows to fit the current column width. If unchecked, the column heading text will be displayed in only one row.

Highlight Key and Required Field Column Headings

If this option is checked, the key fields needed for identifying objects will be highlighted in the case information display. These fields are important if you are planning on pasting data back into Simulator from Excel or reading data in from an auxiliary file. If the necessary key fields are not present in the Excel paste of auxiliary file, the data in those formats cannot be processed by Simulator.

Similarly, the Required Field Column Headings can also be highlighted in the case information display. In order for Simulator to create new objects by reading them from an auxiliary file or an Excel paste, the required fields must be present in the pasted data or the object cannot be created.

Show Header Hints

This option allows the hints for the current field to be displayed when moving the cursor over the column heading.

Copy/Send Options

Typically when you copy information to the clipboard or send data to Excel from a case information display, the first two rows of the copied information contain the type of object the data represents (object name) and the column headings for each column of data. These rows are necessary if you intend to [paste the information back into Simulator](#), but they are unnecessary if you are only exporting data to another program with no intention of pasting the information back into Simulator. Thus, these two options allow you to choose which of the two rows, if either, you wish to have copied along with the actual data from a case information display when pasting in another application.

Disable Auto Refresh

This option prevents Simulator from automatically updating the contents of open case information displays with each solution. If this option is not checked, the data in all open case information displays will be updated automatically to reflect the system state calculated from each Power Flow Solution.

Set Factory Defaults

Clicking on this button will reset the options to their defaults.

Limits Options

These options can be found on the [PowerWorld Simulator Options](#) dialog, and are used for options specific to limit enforcement or identification. Some of these settings are also available in other locations.

Enforce Generator MW Limits

If checked, generator MW limits are enforced.

Enforce Generator Ramp Limits

If checked, generator MW ramp limits are enforced.

Automatically Open Overheated Lines During Simulation

If checked, overheated lines automatically open during simulation.

Highlight Field of Objects with Limit Violations

If checked, the fields for objects on the diagram that correspond to device parameters with range limits that are violated will be colored according to the **High Limit Color** and **Low Limit Color** fields. The colors of these two fields can be defined by the user. This highlighting applies only to the following fields:

- Bus voltage
- Interface MW
- Line MW, Mvar, MVA and Amp
- Generator MW and Mvar
- Load MW
- Switched Shunt Mvar
- Transformer Phase and Tap Ratio

If other highlighting is desired see the more flexible [Dynamic Formatting](#) feature. Using Dynamic Formatting, much more complex highlighting of any field or object can be achieved.

File Management Options

There are two sub-categories on the File Management Options tab: PowerWorld Files, and EPC and RAW Files.

PowerWorld Files

Automatic Loading of Auxiliary File

A default auxiliary file can be loaded when the present case is opened by checking the **Automatically load an auxiliary file when the present case is opened** check box. The auxiliary file given under **Auxiliary File** will then be loaded with the present case. The full path needs to be included with the file name.

Automatic Loading of Auxiliary File with ANY Case

A default auxiliary file can be loaded when each case is opened by checking the **Automatically load an auxiliary file when ANY case is opened** check box. The auxiliary file given under **Auxiliary File** will then be loaded with each case. The full path needs to be included with the file name.

Automatic Archiving of PWB Files

This option allows you to effectively make backup copies of your working case every time you save the pwb file. For example, assume you have a case named Test Case.pwb. With the Automatic Archiving turned on, saving the case will first create a copy of the original file and rename it Test Case_1.pwb. The character used as a delimiter can be chosen optionally. The case with any changes you have just made will then be saved as the new Test Case.pwb file. Each time you save the case, the latest version is named Test Case.pwb, the last Test Case.pwb is renamed with the delimiter and 1 appended, and all other archived versions will be renamed with their number incremented by 1. The number of archive versions to maintain can also be chosen by setting the maximum number of archive files property. The automatically archived cases will be saved in the same directory from which the last case was opened.

Autosave Every ... Minutes

This option allows the user to specify how often (in minutes) the case is automatically saved. A value of zero means that this option will not be used.

Save Unlinked Elements of contingency, interface and injection group records in the PWB file

Checking this box allows saving any unlinked contingency, interface, and injection group records in the PWB case. The unlinked elements are only created when read from an auxiliary file containing unlinked records.

EPC and RAW Files

Display GE Additional Data in Case Information Displays

A large amount of data which is not relevant to the Simulator power flow solution is contained in the GE EPC file. This data is now all read and stored in Simulator and Simulator's PWB file format. By default columns which represent this extra data are not available when choosing [Display/Column Options](#) on a [Case Information Display](#). If you would like these fields to be available, check this box.

Load Additional Data from ECF

Older versions of Simulator did not read all the extra data from the GE EPC file and instead stored it in files with the ECF extension. If you would like to read in one of these old ECF files, click this button.

Remove Additional Data from Case

Because all of the extra data in the EPC can take up a lot of space (approximately doubles the PWB file size and increases the memory foot-print of Simulator itself too), you may want to just delete all this information. Click this button to permanently remove this data from your case.

GE EPC Motor Data Table

Part of the extra data read from the EPC file is Motor Data which does not map to an object used inside PowerWorld. To see this information, click on this button.

PTI RAW Transactions due to tie-line loads

Loads may be assigned to a different area than the load's terminal bus. PTI offers an option that allows the user to ignore this assignment when calculating the tie-line flow for an area.

(AREA INT CODE = 1 FOR LINES ONLY)

(AREA INT CODE = 2 FOR LINES AND LOADS)

Simulator does not allow you to define a load this way and then choose to ignore it. Most RAW files that PowerWorld has seen seem to be solved ignoring the loads that are in different areas than their terminal bus by using the "lines only" option. To overcome this, when PowerWorld reads a RAW file, [MW Transactions](#) are automatically created with the ID "RAW_LOAD". (You can optionally choose to load the RAW file "with options". This will allow you to choose whether or not the transactions should be added. See [Case Formats](#) for more information.) These transactions are between the load's area and its terminal bus and are created so that the export from each area correctly matches a case solved using "lines only". If you know that your case was solved using "lines and loads", then click the button **Remove transaction due to tie-line loads**. Clicking this will delete all the MW Transactions with an ID of "RAW_LOAD". You may also click the button **Add transactions due to tie-line loads** to recreate a list of transaction which represent flows between loads with different areas than their terminal bus.

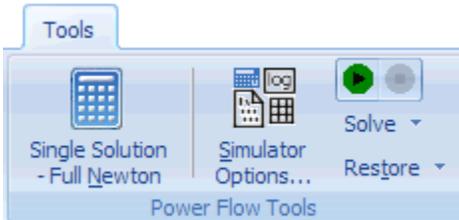
Transformer Regulated Bus Side

EPC and RAW files require the storage of which side of the transformer the regulated bus is on to determine which direction to move the tap in order to regulate voltage. In PowerWorld the tap sensitivity is always calculated when a transformer needs to move so this is not relevant. The buttons are provided to determine this side. If you have a solved power flow solution, then using tap sensitivities is good. If you do not have a solved power flow, then Recalculate using Closest Bus will also work (this just determines how far away the regulated bus is from the FROM and TO bus by summing up the total impedance that must be crossed to reach the regulated bus.)

Power Flow Tools Ribbon Group

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from either the [Quick Access Toolbar](#) or the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information.

The following tasks are also available from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab.



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Simulator Options

When choosing any of the solution methods, the options as specified in the [Simulator Options](#) will affect what calculation is performed. solves the power flow using the either the Full Newton AC load flow or the DC Approximation load flow, as specified in the [Simulator Options](#).

Play Button

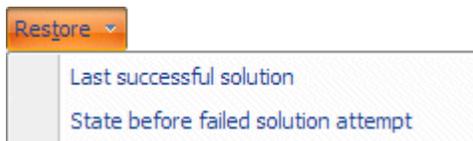
Click this button to cause repeated solutions of the system and to cause the animation of oneline diagrams

Pause/Stop Button

Click this button to stop the action of the Play button.

Restore Menu

The Restore menu offers option to either Restoring a Previous Solution or a Restore a Previous State



Sometimes a power flow attempt won't converge to a solution. When this occurs, the voltages and angles calculated by the solution engine will not satisfy the real and reactive power balance constraints at each bus. Then, the state currently stored in memory will not be an actual system operating point. It is often very difficult to coax the system to solve once it has failed to converge.

To help you recover from a solution attempt that has failed to converge (both timed simulations and single solutions), Simulator offers you two options.

After Simulator solves a system successfully, it will store the voltages and angles it found in memory. If the changes that you then make to the system result in a system that can't be solved, you can select **Restore > Last Successful Solution** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to reload the results of the last converged solution. After reloading this information, Simulator will re-solve the system and refresh all displays.

In addition to restoring the last converged solution, Simulator also gives you the ability to restore the state of the system as it was just prior to the unsuccessful solution attempt. This can be thought of as "un-doing" the effect of the solution attempt. Before attempting a solution, Simulator stores the state of the system in memory. If it solves the power flow successful, Simulator will discard this pre-solution state. However, if the power flow fails to converge, Simulator will keep the state in memory. To recover it, select **Restore > State Before Solution Attempt** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Simulator will replace the non-converged post-solution state with the pre-solution state and refresh the displays. You can then play with the system to try to make it easier to solve.

If you are working with large systems, you should be aware that saving these system states can consume a lot of memory. Therefore, Simulator offers you the option to disable one of both of these features. To do this, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab to open the [Simulator Options Dialog](#), and then go to the Storage sub-tab of the Solution tab. You will see two checkboxes there that can be modified to control whether or not these extra system states are saved.

Solve Menu



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab, or from the [Quick Access Toolbar](#). This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Polar NR Power Flow

This performs a Newton Raphson power flow solution using polar coordinates instead of rectangular coordinates as is normally done in Simulator.

DC Power Flow

This automatically changes the Simulator Options to use the [DC Approximation](#) and then immediately performs the DC power flow solution.

Reset to Flat Start

Select **Solve > Reset to Flat Start** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to initialize the Power Flow Solution to a "flat start." A flat start sets all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero.

Usually, a flat start should be used only if the power flow is having problems converging. You can also use the flat start option on the [Simulator Options Dialog](#) to initialize every solution from a flat start.

Robust Solution Process

The Robust Solution Process provides a method to attempt to reach a solution when the standard load flow (Newton-Raphson) solution fails. The robust process performs a solution in a series of steps.

First, the robust solution will turn off all controls in the case. Then the load flow will be solved using a fast decoupled power flow. If the fast decoupled power flow reaches a solution, Simulator then immediately solves the load flow using the Newton-Raphson load flow, still keeping the controls turned off. If the Newton-Raphson solution is also successful, Simulator will begin adding controls back into the solution process, one type of control at a time. Thus the generator MVAR controls are added back in, and the load flow is resolved. Then the switched shunt controls are restored, and the load flow is again resolved. Simulator will continue in this manner by reintroducing next the LTC control, followed by the area interchange control, and lastly the phase shifter control. Furthermore, when reintroducing the phase shifter control, the controls are added one at a time for each phase shifter, with a load flow solution occurring after each.

Solving the Power Flow

At its heart, Simulator is a Power Flow Solution engine. Power flow is a traditional power engineering calculation that is performed to determine the flows on all lines and the voltages at all buses in the system given the power injections at all buses and the voltage magnitudes at some of them.

The power flow problem entails solving a system of nonlinear equations. Solving a nonlinear system requires the use of an iterative algorithm to hone in on the correct solution. Many nonlinear system solvers have been developed, and PowerWorld provides access to the full Newton-Raphson method with an optimal multiplier and the fast decoupled method.

Usually, the power flow computation converges quickly. However, it is certainly possible to model conditions for which no Power Flow Solution exists, or for which the algorithm cannot converge to the solution within the [maximum number of iterations specified](#). For such situations, the [message log](#) will provide a message indicating that the computation failed to converge. Furthermore, unless [blackouts are disabled](#), the screen is grayed, and a message indicating a blackout has occurred is shown.

In order to calculate the power flow solution, simply press the Single Solution button on the main Program Toolbar. Clicking Single Solution actually performs several pre-processing activities and then runs three nested loops which solve the power flow: MW Control (Outer) Loop, Controller (Middle) Loop, and Power Flow (Inner) Loop.

Pre-Processing Activities

The Power Flow Loop is where the traditional power flow matrix equations are solved.

Angle Smoothing

When a transmission branch status is changed from OPEN to CLOSED across a branch that has a large voltage angle difference, this can introduce a very large initial power flow mismatch and cause the inner power flow loop to diverge. Angle Smoothing is always enabled and will alleviate this large angle difference by smoothing the angles in the system around the newly closed in branch resulting in much better power flow convergence. Angle smoothing will also work if a series of branches are all closed in together. A message will be written to the [message log](#) to indicate this is occurring.

Generator Remote Regulation Viability

Generators are allowed to remotely regulate any bus in the system. However, if there is no transmission path between the generator terminal bus and the remotely regulated bus which does not pass over any other PV buses, then the generator will not be able to regulate this bus remotely. Additionally this will introduce a numerical condition that makes the inner power flow loop not converge. This pre-processing activity ensures catches this condition and prevents these generators from performing voltage regulation. A message will be written to the [message log](#) to indicate this is occurring.

Estimate MW Change Needed

When large changes in generation or load are made to the system, then ultimately this entire mismatch will show up at the island slack buses during the first inner power flow loop. This can cause the inner power flow loop to not converge. When automatic generation control is enabled for this system, then this pre-processing activity will automatically try to initialize generator outputs throughout the system to prevent the entire mismatch from appearing at the slack bus. Eventually the MW Control (Outer) Loop will be executed to bring generators to their proper outputs as specified by the Area or Island AGC choices.

Three Nested Loops

Power Flow (Inner) Loop : Red Loop

The Power Flow Loop is where the traditional power flow matrix equations are solved. There are several [Common](#) and [Advanced](#) Solution options which affect this loop. In the [message log](#), a **RED**

outline will be drawn around the inner power flow loop. Additionally, a **PURPLE** outline will be drawn around any generator Mvar limit checking which occurs inside this loop.

Controller (Middle) Loop : Green Loop

Once the Power Flow Loop is solved, control devices check if their control requirements are being met. Control devices are checked in the following order or precedence.

1. Generator Mvar Limit Checking
2. DC Line Solution
3. Switched Shunt Controls (each switched shunt is examined individually)
4. Load-Tap-Changing (LTC) transformers and Phase Shifter (can be coordinated switching or examined individually)

If any control devices requiring changes, then these changes are made and the power flow loop is re-solved. This continues until no more control loop changes are made. There are several [Common](#) and [Advanced](#) Solution options which affect this loop. In the [message log](#), a **GREEN** outline will be drawn around this loop.

MW Control (Outer) Loop : Blue Loop

After the Control Loop has completed, the MW Control loop is entered and generation (and possibly load) is moved to meet the MW control options set in the case. Normally, MW control is done by [area control](#) with each area varying generation to meet its own load, losses and interchange. However, you may also use [island-based control](#) to dispatch MWs by island. If any MW control is needed, then the Control Loop and Power Flow Loop interaction must be repeated and so on. In the [message log](#), a **BLUE** outline will be drawn around this loop.

A depiction of these loops and how you will see them represented in the message log is shown in the following figure.

Area Loop Iteration #0

4 Inner Iterations

Voltage Loop Iteration #1

3 Inner Iterations

Voltage Loop Iteration #2

3 Inner Iterations

Voltage Loop Iteration #3

2 Inner Iterations

Voltage Loop Iteration #4

Voltage Loop Iteration #5

Area Loop Iteration #1

2 Inner Iterations

2 Inner Iterations

Starting Solution using Rectangular Newton-Raphson

AGC changed area NEVADA slack bus 18259 gen by -500.0MW

AGC changed area NORTHWEST slack bus 40296 gen by 500.3MW

Generation Change Estimate Completed.

Estimation of Generation

Inner Power Flow

Number: 0 Max P: 500.289 at bus 40296 Max Q: 0.038 at bus 44043
 Number: 1 Max P: 268.390 at bus 18259 Max Q: 18.983 at bus 40296
 Number: 2 Max P: 74.898 at bus 18259 Max Q: 18.821 at bus 40296
 Number: 3 Max P: 5.093 at bus 18259 Max Q: 1.870 at bus 40296
 Number: 4 Max P: 0.018 at bus 18259 Max Q: 0.002 at bus 40296

One Voltage Control Iteration Consists Of

Generator Var Limit Checking
 DC Transmission Line Adjustments
 Switched Shunt Mvar Changes
 Transformer Tap Ratio Changes
 Transformer Phase Angle Changes

Inner Power Flow

Gen(s) at bus 18259 at max vars
 Gen(s) at bus 18282 at max vars
 Gen(s) at bus 44271 has backed off var limit
 Gen(s) at bus 45026 has backed off var limit
 Gen(s) at bus 45246 at min vars
 Gen(s) at bus 47860 has backed off var limit
 Gen(s) at bus 66375 has backed off var limit
 DC Line Flow Adjustment
 Switched shunt at bus 40627 moved from 72.2 to 0.0 Mvars
 Calculating Transformer Tap Sensitivities for 31 Transformers
 Transformer 18004 TD 18034 OKT 1 tap ratio moved from 0.99239 to 0.97446
 Transformer 18004 TD 18034 OKT 2 tap ratio moved from 0.99239 to 0.97446
 Transformer 18011 TD 18145 OKT 1 tap ratio moved from 1.01028 to 1.00435
 Transformer 45187 TD 45189 OKT 1 tap ratio moved from 0.95625 to 0.95000
 Transformer 18002 TD 18001 OKT 1 phase moved from 8.1 to 13.2
 Changing Impedance for transformer HA PS (18002) TD H ALLEN (18001) OKT 1 due to impedance correction table
 From 0.00067+j0.04302 to 0.00070+j0.04905
 Transformer 18002 TD 18001 OKT 2 phase moved from 8.1 to 13.2
 Changing Impedance for transformer HA PS (18002) TD H ALLEN (18001) OKT 2 due to impedance correction table
 From 0.00067+j0.04302 to 0.00070+j0.04905
 Taps at 42 transformers adjusted
 Finished voltage control loop iteration: 1

Inner Power Flow

Number: 0 Max P: 432.135 at bus 18001 Max Q: 146.613 at bus 18004
 Number: 1 Max P: 65.168 at bus 18001 Max Q: 38.007 at bus 18259
 Number: 2 Max P: 0.464 at bus 64885 Max Q: 6.826 at bus 18259
 Number: 3 Max P: 0.047 at bus 64885 Max Q: 0.037 at bus 18259

Gen(s) at bus 40344 has backed off var limit
 Gen(s) at bus 40346 at max vars
 Gen(s) at bus 40484 at max vars
 Gen(s) at bus 45026 at max vars
 Gen(s) at bus 45170 has backed off var limit
 Gen(s) at bus 45246 has backed off var limit
 DC Line Flow Adjustment
 Calculating Transformer Tap Sensitivities for 33 Transformers
 Transformer 12076 TD 12077 OKT 1 tap ratio moved from 1.07840 to 1.08462
 Transformer 12090 TD 12091 OKT 1 tap ratio moved from 1.08125 to 1.08763
 Transformer 62062 TD 62061 OKT 1 phase moved from -40.3 to -38.7
 Taps at 18 transformers adjusted
 Finished voltage control loop iteration: 2

Inner Power Flow

Number: 0 Max P: 42.398 at bus 62061 Max Q: 166.372 at bus 40629
 Number: 1 Max P: 0.109 at bus 62062 Max Q: 7.292 at bus 40346
 Number: 2 Max P: 0.008 at bus 62062 Max Q: 0.111 at bus 40627

Gen(s) at bus 45026 has backed off var limit

Check Immediately Mvar Limit Checking

Number: 3 Max P: 0.000 at bus 47087 Max Q: 0.000 at bus 66420

Switched shunt at bus 40877 moved from 10.2 to 0.0 Mvars
 Calculating Transformer Tap Sensitivities for 22 Transformers
 Transformer 18028 TD 18117 OKT 1 tap ratio moved from 0.93859 to 0.94457
 Transformer 40627 TD 40625 OKT 1 tap ratio moved from 0.97230 to 0.98143
 Transformer 40627 TD 40625 OKT 2 tap ratio moved from 0.97230 to 0.98143
 Taps at 3 transformers adjusted
 Finished voltage control loop iteration: 3

Inner Power Flow

Number: 0 Max P: 5.432 at bus 40627 Max Q: 66.306 at bus 40627
 Number: 1 Max P: 0.034 at bus 40625 Max Q: 0.229 at bus 40625

Gen(s) at bus 44271 at max vars
 Gen(s) at bus 45026 at max vars

Check Immediately Mvar Limit Checking

Number: 2 Max P: 0.002 at bus 41213 Max Q: 0.003 at bus 40625

Gen(s) at bus 47290 has backed off var limit
 Calculating Transformer Tap Sensitivities for 20 Transformers
 Finished voltage control loop iteration: 4

Inner Power Flow

Number: 0 Max P: 0.002 at bus 41213 Max Q: 0.031 at bus 47290
 Number: 1 Max P: 0.002 at bus 41213 Max Q: 0.031 at bus 47290

Calculating Transformer Tap Sensitivities for 20 Transformers
 Finished voltage control loop iteration: 5

AGC changed area NEW MEXICO slack bus 10321 gen by 1.5MW

AGC changed area ARIZONA slack bus 15926 gen by 6.8MW

AGC changed area NEVADA slack bus 18259 gen by 7.8MW

AGC changed area SOCALIF slack bus 24004 gen by -6.9MW

AGC changed area NORTHWEST slack bus 40296 gen by -10.0MW

AGC changed area IDAHO slack bus 60100 gen by -7.0MW

AGC changed area MONTANA slack bus 92048 gen by -3.6MW

AGC changed area PACE slack bus 66055 gen by -12.5MW

Generation Adjustment Completed.

Inner Power Flow

Number: 0 Max P: 12.478 at bus 66055 Max Q: 0.002 at bus 26003
 Number: 1 Max P: 0.364 at bus 66055 Max Q: 1.747 at bus 18259
 Number: 2 Max P: 0.004 at bus 66055 Max Q: 0.023 at bus 50982

Calculating Transformer Tap Sensitivities for 18 Transformers
 Transformer 30462 TD 31239 OKT 1 can not effectively control voltage at bus 31239; control will be ignored until the tap sensitivity magnitude incre
 Transformer 39246 TD 36249 OKT T1 can not effectively control voltage at bus 36249; control will be ignored until the tap sensitivity magnitude incre
 Finished voltage control loop iteration: 1

Inner Power Flow

Number: 0 Max P: 0.398 at bus 18259 Max Q: 0.425 at bus 26003
 Number: 1 Max P: 0.000 at bus 16115 Max Q: 0.008 at bus 50985

Calculating Transformer Tap Sensitivities for 20 Transformers
 Finished voltage control loop iteration: 1

Solution Finished In 4.522 Seconds
 Simulation: Successful Power Flow Solution
 Starting Solution using Rectangular Newton-Raphson

Area Control

One of the most important aspects of interconnected power system operation is the requirement that each operating area changes its total generation to match changes in the sum of its load plus losses plus power transactions with other areas. This requirement is normally met by Automatic Generation Control (AGC). The purpose of AGC is to ensure that the actual MW output of an area is equal to the scheduled MW output of the area. The AGC system accomplishes this by first calculating the Area Control Error (ACE), which is defined as

$$ACE = P_{\text{actual}} - P_{\text{scheduled}}$$

In Simulator, $P_{\text{scheduled}}$ for an area is made up of the Area's [MW Transactions](#) and the Area's *Unspecified MW Export*. MW Transactions represent the transfer of power between two areas in the power system. This transaction is done presumably under a contract between the two areas. The advantage of using MW transactions to describe these is that it ensures that the total export of all areas is consistent: if one area is exporting 100 then another area is automatically importing that power. However, each area also specifies a value called the Unspecified MW Export which can be entered on the [Area Information Dialog](#). The unspecified MW export represents an export of power from the area that goes to an unspecified other area. When using unspecified MW exports, it is important that the total unspecified MW exports in the system sum to zero. PowerWorld highly recommends that care be taken when using unspecified exports.

Whenever the ACE is greater than zero, it means that the area is over generating and thus needs either to decrease generation or to sell more. Likewise, whenever the ACE is less than zero, the area is under generating and thus needs either to increase generation or to buy more. AGC works to keep the ACE close to zero.

In Simulator, there are six options for implementing AGC:

No area control

The output of the generators does not change automatically. You must manually change the generation to match system load/losses/transaction variation. All change in load/losses/transaction in this area will be made up at the islands slack bus.

Participation Factor Control

The output of all the area's generators who have their AGC field set to "YES" change automatically to drive the area control error (ACE) to zero. Each generator's output is changed in proportion to its participation factor. Checking this option enables the **Set Factors** button on the [Area MW Control Options](#) tab of the [Area Information Dialog](#), which, when pressed, opens the [Generator Participation Factors Dialog](#). Participation Factor Control only adjusts generation when a change to the system has taken place, such as changing the amount of load in the case, or defining new area to area transactions.

In Participation Factor Control, the ACE is allocated to each AGC generator in the area in proportion to that generator's participation factor divided by the total of the participation factors for all AGC generators in the area. A generator's participation factor cannot be negative. By default, a generator's participation factor equals its current MW setpoint value, but individual participation factors can be changed.

Economic Dispatch Control

The output of all AGC generators in the area changes automatically to drive the area control error (ACE) to zero. Each generator's output is changed so that the system is dispatched economically, based on cost information entered for the generators in the case. Note that cost data is not generally included in standard load flow data. Without realistic cost data entered into Simulator, the

use of the economic dispatch algorithm may not be very useful. Cost data must be obtained from another source and entered into a case in Simulator, either manually or through the use of [Simulator Auxiliary Files](#).

With Economic Dispatch (ED) Control, Simulator tries to change the output of the area's AGC generators economically so that the area's operating cost is minimized. ED control recognizes that some generators are less expensive than others and tries to use the least expensive generators to the largest extent possible.

To do economic dispatch, we need to know how much it would cost to generate one more MW at a particular generator. This is known as the incremental or marginal cost. For example, for the cubic cost curve model, the incremental cost for each generator is modeled using the formula:

$$\lambda_i = IC_i(P_{gi}) = (b_i + 2c_i P_{gi} + 3d_i (P_{gi})^2) * \text{fuelcost} \quad \$/\text{MWH}$$

The plot of $IC_i(P_{gi})$ as a function of P_{gi} is known as the incremental-cost curve. The economic dispatch for a system occurs when the incremental costs for all the generators (λ_i) are equal. This value is known as the system λ (lambda) or system incremental cost. Its value tells you how much it would cost to generate one more MW for one hour. The system lambda becomes important when trying to determine whether or not an area should buy or sell power. For example, if an area can buy power for cheaper than it can generate it, it might be a good idea for the area to buy power.

Area Slack Bus Control

Only the output of the area's slack bus changes automatically to drive the area control error (ACE) to zero. This type of generation control is usually only good for small disturbances to the injections and/or transactions in a case, and can often fail to find a solution when larger disturbances are examined.

Injection Group Area Slack Control

Only the Injection Group specified as the Injection Group Area Slack will change to automatically drive the area control error (ACE) to zero. This allows you to be very specific with each area as to which generation (or load) should vary to maintain ACE.

Optimal Power Flow (OPF)

The [OPF](#) option will only be available if you have the OPF add-on for PowerWorld Simulator. The OPF control is very similar to the Economic Dispatch control in that it attempts to dispatch generation to minimize costs. The additional function of the OPF is to minimize the costs while also obeying line, transformer, and interface limit constraints. This option is also not useful without realistic generator cost information, which usually must be obtained from another source and entered into Simulator to augment a load flow case.

The OPF control also relies on the cost curve in order to perform an economically optimal power flow. However, the OPF routine makes use of piecewise linear curves in its solution algorithm. This does not prevent you from entering the cost information as cubic cost models, described by the equation above. Rather Simulator's OPF routine allows you to specify how to break up the cubic curve and model it as a piecewise linear curve for the OPF algorithm.

In addition, you can also enter piecewise linear curves directly instead of the cubic cost curve models. In fact, a mixture of piecewise linear and cubic models is acceptable. For the economic dispatch routine, whichever type of model is entered will be used directly for each generator. For the OPF routine, all piecewise linear curves entered directly will be used as is, and any cubic models entered will be converted to piecewise linear curves internally during the processing of the OPF algorithm.

Set Generator Participation Factors

Participation factor control is another of Simulator's mechanisms for distributing an area's responsibility to serve its load, losses, and interchange. It is particularly well-suited to implementing [automatic generation control \(AGC\)](#) when you do not have good economic information for an area's generators. With participation factor control, the amount of power that each generator contributes to meeting its areas load, loss, and interchange responsibilities is controlled by the size of its participation factor. The unit that has the largest participation factor contributes the most, and the unit that has the smallest participation factor contributes the least.

The **Set Generator Participation Factors Dialog** gives you a convenient way to define the participation factors for multiple generators. You can set the participation factor according to a number of different formulae and then apply this prescription to all generators in a specific area, all generators in a specific zone, all generators in the system, or all generators whose display filters are currently set to true.

To display the **Set Generator Participation Factors Dialog**, you first need to open the [Area Information Dialog](#) and switch to the Area MW Control Options page. The Area Information Dialog has a button labeled **Set Participation Factors** that is enabled only if the Participation Factor Control option is selected under the Area Control Options heading. Set the area on participation factor control by selecting the Participation Factors option, and then press the **Set Participation Factors** option.

The Set Generator Participation Factors Dialog is divided into two parts. The first part, which occupies the top half of the form, allows you to indicate how the participation factors should be calculated or set for each generator. Your options include:

Max MW Rating of Generator The participation factor for each generator is set to the generator's maximum MW capability.

Difference Between Max and Current Output The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute.

Constant Value of The participation factor for each generator is set to the same hard-coded value.

File The participation factor for each generator is read from a file. The first line of the file should contain the keyword NUMBERS or NAMES indicating whether generators are identified by bus number or by bus name in the file. All subsequent lines should be comma-delimited and contain three fields: the number or name of the generator's bus, the generator's id, and the generator's participation factor.

If you choose any of the first three options, you then must tell Simulator to what generators you want to assign the participation factors. To assign the participation factors to all generators in a specific area, select the **All Generators in Area** option, and then choose the area from the adjacent dropdown box. If you want to assign the participation factors to all generators in a specific zone, select the **All Generators in Zone** option, and then choose the zone from the adjacent dropdown box. If you want to assign the participation factor to all generators in the system regardless of their area or zone affiliation, select the **All Generators in System** option. Finally, if you want to assign the participation factor to just those generators whose display filter criteria evaluates to true, choose the **All Generators With Valid Display Filters** option.

If you instead chose to read participation factors from a file, only those generators whose factors you read from the file will have their factors set by this action. However, unless each generator's associated area is set to control generator output using participation factor control, this information will be ignored. To make sure that each generator's area is set to participation factor control, check

the **Set Corresponding Areas to Participation Factor Control** box. Then, each corresponding area will be set to participation factor control.

Transaction Dialog

The Transaction Dialog can be used to modify or create Base Interchange Transactions between two areas. This dialog can be opened by right-clicking in the Base Interchange table of the [Area Information Dialog](#) and choosing Show Dialog to see an existing transaction definition, or Insert to add a new transaction.

The transaction dialog is divided into two pages of controls:

Information

Transacting Area

This is the "from" area for the transaction. For an export from this area, the transaction value will be positive. For an import into the Exporting Area, the transaction value specified would be negative. Flow out (export) of the exporting area is always considered positive.

Transaction To Area

This is the "to" area for the transaction.

Transaction ID

New in Simulator version 10 is the ability to have multiple transactions defined between the same two areas. Because of this, it is now required that transactions also have a Transaction ID.

Rename Transaction ID

If you wish to change the transaction ID for a particular transaction, enter the new value in the **Transaction ID** field, and press this button.

Switch Directions

Press this button if you wish to reverse the defined Exporter and Importer for the transaction.

Transaction MW Amount

The MW amount of the transaction being defined. This value should be positive for an export from the Exporting area to the Importing area. The value can also be entered as negative to define a transaction into the Exporting area from the Importing area.

Transaction Minimum MW

The minimum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Transaction Maximum MW

The maximum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Exports/Imports Transmission Charge

The cost to transfer power, in \$/MWh. This adds an economic penalty for making the transfer, making the transfer less likely to take place. Half the charge is assigned to the buyer and half to the seller.

Transaction Enabled

Transaction can now be defined and either enabled or disabled. Any disabled transactions will be ignored in both a standard power flow solution and an OPF solution.

Transaction Dispatchable in OPF

Checking this box enables the transfer to be dispatched by the OPF algorithm. Dispatching the transaction makes the two areas of the transaction appear to be one area for the purpose of economically dispatching the generation in the two areas. The transaction can have a maximum and minimum transfer amount when dispatchable, and a transmission charge associated with the transaction.

Determine Price in OPF

Checking this box allows the OPF algorithm to determine the cost associated with the transfer. The cost is determined by the marginal cost of enforcing the power balance constraint for the combined areas. This is the typical way to implement a transfer if *both* areas are on OPF control. If only one of the two areas are on OPF control, then the area which is off of OPF control needs to specify a price for the transfer. This is done by explicitly defining a piecewise linear cost curve.

Piecewise Linear Transaction Cost Curve

These two curves are only enabled if the option **Transaction is Dispatchable in OPF** is checked and the option **Determine Price in OPF** is unchecked. These two curves can be defined for the purpose of assigning a price to the transfer of power between one area on OPF control and another area which is not on OPF control. Separate curves can be defined for export transactions (from the Exporter to the Importer) and import transactions. To add points to the curves, simply right-click in the grid and choose Insert from the local menu. Enter the MW value and corresponding marginal cost for the inserted breakpoint of the piecewise linear curve you are defining. To delete a point, right-click on that row in the grid and choose Delete from the local menu.

Custom

The Custom page of the Transaction dialog is simply a location to log information about the transaction in the Memo box, and to see the values (if any) stored in some of the custom fields for the transaction. To log information about the transaction, simply switch to the Custom page on the dialog, and start typing your information or comments about the transaction in the page, or enter custom values in one of the custom fields.

Calculate MW-Distance

Simulator can estimate MW * Distance quantities for the system's areas and zones that result from a specified transaction. Given a transaction from a specified source to a specified sink, Simulator uses [power transfer distribution factors \(PTDFs\)](#) to estimate the change in flow for each line in the system that results from the transaction. For each line, multiplying the line's change in flow by its length then gives the MW * Distance index for that line. Simulator then sums the MW * Distance indices by area and by zone to obtain the total MW * Distance for each area and zone in response to the specified transaction.

Because the MW * Distance calculations use PTDFs, you must access the MW * Distance functionality from the PTDF Dialog. Once you have calculated PTDFs for a particular transaction by pressing the **Calculate PTDFs** button, click the **Calculate MW * Distance** button to bring up the **MW * Distance Calculations Dialog**.

The top portion of the MW * Distance Calculations Dialog is used to set the lengths of the lines in the case. Although line length is represented as a data element in the power flow case, it often is left blank. However, Simulator needs line length information if it is to calculate MW * Distance indices. Simulator offers a few options regarding the source of line length information. If you do not have access to line lengths, either from the existing case or an external text file, Simulator can estimate line lengths for you. It does this by using the Ohms/Length values you specify in the table for lines of various kV. Simply indicate the voltage levels in the first row of the table, and the corresponding ohms or reactance per length in the second column. You do not need to differentiate here between English and metric units, because the calculation is independent of the measurement system. If you want the length estimates calculated using this table to overwrite any line lengths that may already be present in the case, be sure to check the **Always Estimate Length** checkbox; otherwise, the new estimates will set the lengths only of lines whose pre-defined length isn't greater than zero. If you want the estimates to populate the lengths of lines in the model so that, when you save the model, the estimated lengths are saved as part of the model, check the **Save Estimates With Case**. (This provides a handy way to set line lengths for a case that might not have any defined.) Note that, in performing these estimates, transformers are defined as having zero length. If you do not want Simulator to estimate line lengths but instead want to use the line lengths that are currently stored in memory, check the **Do Not Use Length Estimates** box. Finally, if you want to load line lengths from a text file, click the **Load Line Lengths from File** button. This file can be either comma- or space-delimited, and each line must have the following fields in the order specified:

```
From_Bus_Number To_Bus_Number Circuit_ID Length
```

Once Simulator knows how to calculate line lengths, it can calculate MW*distance indices for each area and zone. Specify the amount of MW that will be transacted in the **Size of Transaction** textbox. You may use the arrows to increase or decrease the size of the transaction. Simulator assumes that the transaction is to occur between the source and sink groups for which you just calculated PTDFs. Press the button labeled **Calculate** to compute the indices. Two tables are populated with the results of the calculation, one for areas, and another for zones. Use the tabs to switch between the two tables. These tables are [Case Information Displays](#) and thus share common to all case information displays. Thus, you can sort the tables, add or delete columns, access the area and zone dialogs, print the tables, and save their content as HTML.

Several options can be set to customize the calculation of MW*Distance. These options are reached from the [MW*Distance Options Dialog](#).

MW-Distance Options

The calculation of MW*Distance quantities can be customized in a number of ways. These options are set from the MW*Distance Options Dialog.

Include Tie Lines Only

If this box is checked, then the only branches that contribute to the MW*Distance calculation are those that tie two areas together. Otherwise, both tie lines and lines internal to areas and zones are included in the calculation. In the latter case, tie lines are assumed to belong to the area that owns the metered end of the branch.

Internal Flows

If you choose to include both tie line flows and flows internal to areas and zones in calculating MW*Distance quantities, you have two options for how to treat internal flows. You can ignore flows resulting from the transaction that flow in the reverse direction of the existing flow on a branch by checking the **Include flow increases only** checkbox. You can also choose to treat all such counterflows as negative contributions to an area or zone's MW * Distance value by checking the **Deduct flow reductions** checkbox.

Omit Branches

To omit branches for which the PTDF corresponding to the transaction is less than a specified value, specify a nonzero percentage in this textbox.

Area Control Error (ACE) Chart

The ACE chart plots the area control error for an area over time. For details on ACE, please see topic [Area Control](#). To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > ACE Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to plot the data when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area Load and Generation Chart

The Load and Generation chart plots an area's load + losses and generation in MW when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Load and Generation Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Load/Generation MW axis) by right-clicking anywhere on the axis itself and specifying the axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area Losses Chart

The Area Losses chart plots an area's real power losses when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Losses Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Load/Generation MW axis) by right-clicking anywhere on the axis itself.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area MW Transactions Chart

The Scheduled MW Transactions chart plots the scheduled real power (MW) transactions for an area when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area MW Transactions Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area Average Cost Chart

The Average Cost per MWH chart plots the average cost per MWH for an area over time. This value is calculated by dividing the total cost of operating the area (generation cost + purchased power cost - revenue from power sales) by the MW load in the area. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Average Cost Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to plot the data when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals. Note that as the area's load increases, the average cost per MWH tends to increase.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Bus View Display

The **Bus View Display** feature provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus. The bus view displays enable convenient bus-by-bus navigation through the power system. From the bus view, you can find out a bus' voltage and angle, the load, shunt compensation, and generation connected to the bus, and the flows on all lines emanating from the bus. You can also discover the bus' area and zone affiliations, as well as the bus' marginal cost. Moreover you can find out all information about the elements associated with the bus by directly invoking their associated information dialogs. The advantage of the bus view displays, is that they are auto-created oneline diagrams.

Along the top of the bus view display resides a panel of controls. The buttons labeled **Back** and **Ahead** allow you to step through the history of buses you have viewed thus far. To move the bus view to a particular bus, type a number or name into the edit box next to the caption **Bus**. If a number is typed the case will be scanned for a bus of that number, otherwise the case will be scanned for a bus that starts with the string entered. If you're unsure which bus that you are looking for click the **Find...** button to bring up a display that allows you to search using wildcard strings for the bus. The **History** menu will show a list of the last 30 buses which have been visited in a bus view. There are also drop-downs in this top panel for **Options** and **Views**. These are discussed below.

Below this top panel sits the actual bus display. The bus you have chosen to inspect, which we shall refer to as the *target bus*, is represented by a long, thick horizontal line. Notice that the bus' voltage in kV and per unit, its angle, and its marginal cost are specified to the left of the bus. Any loads and generating loads connected to the target bus are drawn above the bus symbol, along with their associated annotation. Also any radial connections to a single bus containing load, generator or switched shunts will be drawn above the bus symbol.

Emanating from the bottom of the bus symbol are all transmission lines and transformers that connect the target bus to its neighbors. The transmission line and transformer symbols are equipped with pie charts and annotation identifying flows as measured at the target bus, as well as arrows to identify the direction of MW flow on the branch. Neighboring buses are represented as filled rectangular regions, with symbol indicators included if other types of devices, such as loads, generators, etc., are attached. When you drag the mouse over one of the neighboring bus symbols, it turns into a pointing finger. Clicking the left mouse button when the mouse cursor is in this shape redefines the target bus to be the bus whose symbol you just clicked. The bus view display is redrawn to show the same sort of display for the newly chosen target. You can go back to the previously displayed target bus by clicking the **Back** arrow, and then return to this record by clicking the **Ahead** arrow.

It is useful to think of the bus view displays as nothing more than an additional oneline diagram. In other words, you interact with the objects drawn on the bus view display in the same way you work with objects on a more conventional Simulator oneline. Right-clicking on any power system object will bring up that object's local menu, which includes a link to the object's associated information dialog. As on a conventional oneline diagram, flows on a bus view display can be animated. Right-clicking on the bus view display's background will generate the same local menu as other oneline diagrams.

The bus view display can be generated using any of the following methods:

- Go to the [Case Information](#) ribbon tab, and click on the **Bus View** from the **Views** ribbon group
- Go to the [Onelines](#) ribbon tab, and click on the **Bus View** from the **Views** ribbon group.
- Right click on the bus of interest on the oneline diagram to display the bus' local menu, and choose Bus View. The bus view display will open with the selected bus already displayed.

- From any of the case information displays that convey bus information, right click on a record to bring up its local menu, and choose Bus View Oneline. The bus view display will open with the corresponding bus already displayed.
- Click the **Bus View** Button on the [Case Information Toolbar](#).

To switch between the bus view and the main oneline, use the **Open Windows Menu** available on the **Views** ribbon group on the [Case Information](#) ribbon tab, [Case Information](#) ribbon tab, or [Windows](#) ribbon tab. To close the bus view display, simply close the form using the **X** button in the top right corner of the bus view display.

Bus View Options

Layout Options

Show Consolidated Super Bus Topology

Only available with the Topology Processing Add-on. This option will be available if at least one branch in the system has been marked as Consolidate Branch = YES. Checking this option will convert the bus view so that it only represents Super Bus groupings and does not show the all the individual buses that have been consolidated to create a Super Bus.

Show Only Breaker for Consolidate Branch

Only available with the Topology Processing Add-on. For branches which are marked as Consolidate Branch = YES, only one circuit breaker will be drawn on the bus view.

Show Serial Buses

When putting in the branch connections, the Bus View display will look out into the network and find the next bus which has more than two neighbors. It will then make this the destination bus for that branch of the Bus View. The intermediate buses will then be shown in order above the destination branch. This option works especially well in systems with a lot of multi-section lines.

Show Equivalent Lines

This option indicates to the Bus View display whether or not to include branches representing equivalent circuits as connections in the display.

Represent Multi-Section Line Objects

This option indicates to the Bus View display whether or not to draw multi-section line display objects when appropriate. If this option is not checked, then the intermediate buses of the multi-section lines will be rendered normally. If this option is checked then the intermediate buses of multi-section lines will not be rendered on the bus view and only the terminals of the multi-section line will appear.

Show Field Suffixes

This option specifies whether to display the field values units as suffixes. If this option is not selected, all the fields will be display as a value without units.

Include Field Labels

Selecting this option will place labels for each displayed field on the bus view diagram.

Default Drawing Values

Choosing this option will open the Default Drawing Values for New Objects dialog. Changing these options can change some of the drawing aspects of the bus view, including device color and font size or color.

Dynamic Formatting

Click on this to bring up the [Dynamic Formatting](#) for case information displays, and all views and onelines.

Number of Tiers

The bus view can display one or two "tiers" of buses in the display. Use this selection from the **Options** menu to toggle the number of tiers displayed.

Open Multiple Bus Views

This option indicates whether to open multiple bus views simultaneously. Choices are never, always, and prompt for confirmation when a new additional bus view is about to open.

Show Hints

When this option is checked, holding the cursor over an object will briefly pop up a hint box containing information about that object.

Views

Define Custom View

The fields displayed on the bus view can be customized using this option. Clicking on this option will open a customization settings display, in which you can add and remove field definitions for the objects on the bus view display. Customized bus view layouts can be saved with the case for recall, identifiable by a custom bus view layout name. Custom layouts can also be saved to a file for loading into another load flow case.

Input Data

Switching the bus view to Input Data changes the bus view from displaying system state information to displaying input data information. For example, switching to Input Data view will display line impedances and limits, generator minimum and maximum outputs, etc. The default Input Data view fields can be modified using the Define Custom View customization dialog.

System State

Switching the bus view to System State changes the bus view from displaying Input Data information to displaying system state information. This will result in line flows being displayed, voltage and angles displayed, etc., of the current solution state of the system. The default System State view fields can be modified using the Define Custom View customization dialog.

Substation View Display

The Substation View Display feature is analogous to the [Bus View Display](#).

Along the top of the substation view display resides a panel of controls. The buttons labeled **Back** and **Ahead** allow you to step through the history of substations you have viewed thus far. Substation View options are available from the **Options** menu. To find a substation using search tools, click on the **Search For** button. The next two controls following the Substation label allow to specify a substation name (in the first text box) or a number (in the second text box). If you type a number or name that does not exist, the substation display will continue displaying the current substation.

Below this top panel sits the actual substation view display.

Just as with the bus view, it is useful to think of the substation view displays as nothing more than a oneline diagram. In other words, you interact with the objects drawn on the substation view display in the same way you work with objects on a more conventional Simulator oneline. Right-clicking on any power system object will bring up that objects local menu, which includes a link to the object's associated [information dialog](#). As on a conventional oneline diagram, flows on a substation view display can be animated. Right-clicking on the substation view display's background will generate the same [local menu](#) as other oneline diagrams. The substation view can be generated using any of the following methods:

- Go to the [Case Information](#) ribbon tab, and click on the **Substation View** from the Views Ribbon Group
- Go to the [Onelines](#) ribbon tab, and click on the **Substation View** from the **Views** ribbon group.
- Right click on the substation of interest on the oneline diagram to display the substation's local menu, and choose Substation View. The substation view display will open with the selected substation already displayed.
- From any of the case information displays that convey substation information, right click on a record to bring up its local menu, and choose Substation View Oneline from the Substation Records menu. The substation view display will open with the corresponding substation already displayed.
- Click the **Substation View** Button under the **Records** menu on the [Case Information Toolbar](#).

To switch between the substation view and the main oneline, use the **Window** menu tree on the main menu. To close the substation view display, simply close the form using the **X** button in the top right corner of the substation view display.

Chapter 10: Run Mode Tools and Options

Run mode provides a number of commands for simulating and modifying the case that are different from Edit Mode.

This chapter covers the following:

- General Tools
- Contingency Analysis
- Fault Analysis
- Contouring
- Distribution Factors
- Sensitivities

Generator Economic Curves

Four characteristic curves describe the efficiency and resulting costs associated with operating a particular generating unit. These four curves plot

- Fuel Cost
- Heat Rate
- Input-Output
- Incremental Cost

Simulator can display plots of all these curves. To display a particular plot for a generator, right-click on the generator in Run Mode to display its local menu, and then select the plot you wish to see. The plot will be presented in its own window. The windows for all plots exhibit identical characteristics. For example, the current operating point is identified by a red filled circle. Right clicking on an open area of the window displays the plot's local menu which allows you to print the plot, save it to a file, or copy it to the clipboard for use in other programs. To adjust the length and number of intervals shown on an axis, right-click on the *axis* (not on the *numbers*) then specify the min and max display values and number of intervals. To close a plot window, simply click the X button in its top right corner.

The Run Mode generator local menu also provides access to a fifth type of plot curve - the "All Area Gen IC Curves" plot. This plot simply shows the incremental cost curves and present operating points of all generators in the same area as the generator on which you clicked.

Fuel Cost Curve

The fuel cost curve specifies the cost of fuel used per hour by the generating unit as a function of the unit's MW output. This is a monotonically increasing convex function.

Heat-rate Curve

The heat rate curve plots the heat energy required per MWH of generated electrical output for the generator as a function of the generator's MW output. Thus, the heat rate curve indicates the efficiency of the unit over its operating range. Generally, units are least efficient at the minimum and maximum portions of their MW output capability and most efficient somewhere in the middle of their operating range. The vertical axis is plotted in MBtu/MWH and the horizontal axis is plotted in MW. You may interpret the heat rate for a generator producing X MW as follows: the heat rate indicates the amount of heat input energy per MWH of generation required to produce X MW of power. The lower this number, the less input energy is required to produce each MWH of electricity.

Input-Output Curve

The input-output curve is derived simply from the heat-rate curve by multiplying it by the MW output of the unit. This yields a curve showing the amount of heat input energy required per hour as a function of the generator's output.

Incremental Cost Curve

By multiplying the input-output curve by the cost of the fuel in \$/MBTU, one obtains the cost curve for the unit in \$/hr. By taking the derivative of the cost curve, one obtains the incremental cost curve, which indicates the marginal cost of the unit: the cost of producing one more MW of power at that unit.

Find Branches that Create Islands

To access the Find Branches that Create Islands, go to the [Tools](#) ribbon tab, and choose **Connections > Find Branches that Create Islands** from the [Other Tools](#) ribbon group.

There are several options for selecting which ac lines to process.

Line Processing Options

All ac Lines

All ac lines in the power system model will be processed.

Use Area/Zone/Owner Filter

All ac lines that meet the defined Area/Zone/Owner Filters will be processed.

Select Area/Zone... to display the Area/Zone/Owner Filters dialog.

Use Selected

All ac lines that have the Selected? field set to 'YES' will be processed.

Click Select Lines... to display all ac lines and change the Selected? field.

Meets Filter

All ac lines that meet a selected advanced filter will be processed.

Use the drop down box to select a defined [advanced filter](#) or click Define Filter to display the Advanced Filters for Branch Dialog. This dialog will allow you to define a new advanced filter for a branch or update an existing filter.

By checking **Do not display radial lines creating a single bus island** those lines that only island a single bus will not be displayed with the results.

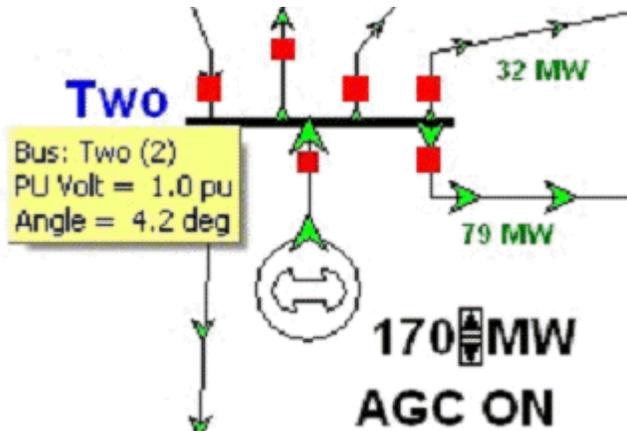
Click **Determine Branches** to start the processing once all options have been set.

Because the processing of ac lines in a large power system may take some time, there is an **Abort** button that is enabled once the line processing has started that will stop the processing at any point.

The list of resulting ac lines that create islands will be displayed under **Branches that Create Islands**. To show the list of buses that are islanded by an outage of any line in the list, select a line and the **Islanded Buses** list will be populated. Both the Branches that Create Islands list and the Islanded Buses list are [case information displays](#) and have the same [local menu options](#) and characteristics of other case information displays.

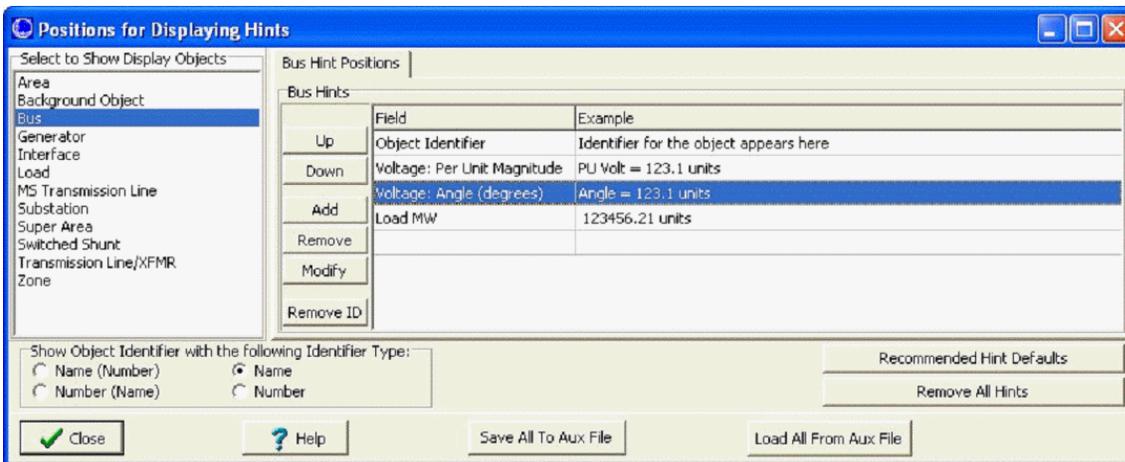
Custom Hint Values

Simulator can be configured to show a short hint string when the mouse is hovered over objects on the oneline diagrams. This hint string will appear in a yellow box as shown below.



This option must be set first by opening the [Simulator Options dialog](#) and then going to the **Online** category and finally checking the **Show Online Hints**. The user may then customize what text and values appear in the hint by choosing **Custom Hint Values** on the **General Options** ribbon group of the [Options](#) ribbon tab. These **General Options** are also available on the [Online](#) ribbon tab.

This brings up a dialog on which customizations are made for each type of display object that can show a hint.



Choose the type of object on the left side of the dialog. With this display object chosen, choose which fields of this dialog you would like to include as part of the custom hint. Click the **Add**, **Remove**, or **Modify** buttons to add, remove, or modify a field to the hint. Click the **Up** and **Down** buttons to change the order in which the fields appear. On the dialog that appears after clicking Add or Modify, you may customize the field with a prefix, modify the digits and decimal points, and specify whether to show the units as a suffix. The Example column shows an example of what text will appear for the hint.

The *Object Identifier* is also available that will appear as the first line of the hint if chosen. To remove this identifier, click the **Remove ID** button. When the object identifier is not being shown the **Remove ID** button caption will change to **Add ID**. Click the **Add ID** button to add the object

identifier back to the hint. You may specify the format of the object identifier by setting the **Show Object Identifier with the following Identifier Type** option appropriately.

The hint customizations are stored with the PowerWorld Binary file (PWB) and in the Windows Registry. The customizations affect all oneline diagrams (including the Bus View and Substation View). The customizations may also be stored in an [Auxiliary File](#) for use in moving the customizations between computers and cases. Click the **Save All to Aux File** button to save all these customizations to an Auxiliary File.

Line Loading Replicator

The Line Loading Replicator will attempt to set the loading on a transmission line or interface to a desired real power flow amount. This is accomplished by calculating distribution factors for a selected group of loads and generators to determine how the real power injection of these elements can be adjusted to produce the desired flow. If the desired flow cannot be met, the new flow amount that can be achieved and the injection changes required to meet this flow will be reported. Minimum and maximum MW limits for loads and generators will be enforced when calculating the injection changes.

To open the Line Loading Replicator, go to the [Tools](#) ribbon tab, and choose **Line Loading Replicator** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

Select Device

Device Type

Select whether to adjust the flow on a transmission line/transformer or an interface. All devices must already be defined in the case.

Device Identifier

Specify the near bus, far bus, and circuit identifier for a branch or the interface name and number for an interface. Branches will be monitored in the near bus to far bus direction and interfaces will be monitored in the direction in which they are defined to be monitored.

Present Flow

Present real power flow on the selected device.

Desired Flow

Desired real power flow on the selected device. The difference between the Desired Flow and the Present Flow (Desired Flow – Present Flow) will be the flow change for which injection changes will be calculated.

Available Injection Groups

Injection Group Identifier

Specify the injection group name and number that contains the loads and generators whose injection can be changed to reach the Desired Flow. The injection group must already be defined in the case. The injection group definition is used simply for selecting the group of elements to be used in the calculations; the participation factors defined with the participation points in the injection group are not used in the Line Loading Replicator tool.

Only Include AGCable Generation and Load

If checked, only those loads and generators in the selected injection group whose AGC status is *YES* will be included in the calculations.

Calculation Method

Select the method used to calculate the distribution factors used to determine real power injection changes.

Max and Min Load Limits for Injection Changes

Use Max and Min Load Values

If checked, the maximum and minimum load values will come from the maximum and minimum values defined with each individual load record.

Set Max and Min Load Values

Click this button to show the [Load Records Case Information Display](#). The maximum and minimum load values can be defined by displaying the **Max MW** and **Min MW** fields and setting the values as appropriate.

Use Multiplier on Present Value

If checked, the minimum and maximum value for each load will be determined by applying the **Min Multiplier** and **Max Multiplier** to the present load value.

Calculate Injection Changes

Click this button to calculate the injection changes required to meet the Desired Flow on the selected device.

Injection Changes

This is a case information display that will provide a summary of the injection changes needed, and the elements that can effect these changes, in order to meet the Flow Achieved on the selected device. The Flow Achieved will differ from the Desired Flow if there are not enough injection changes available to meet the Desired Flow.

The results are reported in terms of injection. For generators, a positive injection means an increase in generator output and a negative injection means a decrease in generator output. For loads, the opposite is true; a positive injection means a decrease in load and a negative injection means an increase in load.

Injection Change, **Present Injection**, and **New Injection** fields will be shown by default. These Present Injection and New Injection fields are both calculated based on the present injection of the given element with the New Injection value also being dependent on the Injection Change value. The Present Injection and New Injection values will be updated any time that the present injection for an element changes in the power system. This is important to keep in mind if making any changes to the power flow case while the Line Loading Replicator dialog is open.

Injection changes are calculated so that generator and load limits are enforced. Which load limits are used is determined in the **Max and Min Load Limits for Injection Changes** option settings. The generator limits used are the minimum and maximum limits defined with each generator. These limits can be accessed from the [Generator Case Information Display](#). The load and generator limits can also be displayed in the Injection Changes table by adding the **Min MW** and **Max MW** fields to the table.

Total Injection Increase/Decrease

This value is the amount of injection increase and decrease required to meet the Flow Achieved. The net injection will be zero.

Flow Achieved

Flow amount that can be achieved by implementing the Injection Changes listed. This value will differ from the Desired Flow if there are not enough injection changes available to meet the Desired Flow.

Implement Injection Changes

Click this button to make changes to the power flow case to match the calculated Injection Changes. Automatic generation control (AGC) and phase shifter control are disabled for the entire case when choosing to implement changes.

Implement Injection Changes and Solve Power Flow

Click this button to make changes to the power flow case to match the calculated Injection Changes and then solve the power flow. The power flow is solved with the settings that are specified in the [Power Flow Solution Options](#) specified with [Simulator Options](#). When choosing to implement

changes, these options are updated so that automatic generation control (AGC) and phase shifter control are disabled globally for the case. The Present Flow value is updated after the completion of the power flow solution.

Dynamic Formatting Overview

The [Dynamic Formatting dialog](#) will allow the specification of how a graphical object will be rendered depending on the power system object that is representing. The graphical object refers to objects in the oneline diagram, in bus and substation views, as well as to some parameters of the case information displays.

The case and the one-line diagrams will have each a list of Dynamic Formatting settings. The Dynamic Formatting settings defined in the case are always applied to bus and substation views, and optionally they can be applied to the case information displays. The one line diagrams will use its own settings, but optionally can use the general settings defined with the case.

In general, the oneline dynamic formatting settings have a higher priority, followed by the case dynamic formatting settings, and this can't be reversed. Also, inside each of the list of dynamic formatting settings, a priority can be specified, so that objects can be rendered according to the settings with the highest priority.

In the case of the oneline diagrams, the dynamic formatting settings will only be applied during Run Mode. However, these settings can be modified at any time, without regard for the mode.

Dynamic Formatting Dialog

The Dynamic Formatting dialog for the active oneline can be accessed by:

- Selecting **Dynamic Formatting** >  **Active Oneline** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.
- Selecting **Dynamic Formatting (Active Oneline)** from the [oneline local menu](#), or

The Dynamic Formatting dialog for the general case information displays, bus and substation views, and for all onelines can be accessed by:

- Selecting **Dynamic Formatting** >  **Case Info / All Views And Onelines** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.
- Selecting **Dynamic Formatting (All Views)** from the [bus view local menu](#) or the [substation view local menu](#), or

This dialog presents the following options:

Set Format Priority

It is possible to have multiple dynamic format definitions created and stored with a case and/or active oneline diagram. This option allows you to set a format priority order for all of the dynamic format definitions, so that any diagram objects that meet more than one dynamic filter will use the settings of the dynamic filter with the highest set priority.

Allow Oneline to use dynamic formatting defined with case

This option will only be available when the dynamic formatting settings correspond to an active oneline. It indicates whether or not the oneline diagram will use the dynamic formatting definitions specified in the case.

Object Type

The type of power system object with graphical representation, such as a bus, a load, a generator, etc.

Filter Criteria

The filter that applies to the object type defined. The Dynamic Formatting settings will apply to the graphical object only if the corresponding power system object meets the specified filter. If the criteria box is empty, the Dynamic Formatting settings will assume that all the objects of the specified type meet the criteria.

Formatting Active

If this option is unchecked, the dynamic formatting definition will be ignored when the objects are rendered. Otherwise, it will be applied if there are graphical objects whose characteristics match the rest of the characteristics defined in this dialog.

Force visibility

When this option is checked, the objects will be displayed (assuming the dynamic formatting applies to them) independently of the visibility of the layer to which such objects belong, and with no regards of the low and high zoom levels.

Show Lookup Tables

Checking this box will display the **Use Lookup** options with the characteristic settings. The **Use Lookup** option with certain field characteristics allow you to define a table of different field values and corresponding characteristic values. If you check Show Lookup tables, and then check the Use Lookup field next to one of the characteristics, you will open the Lookup definition table. This table allows you to select a field for the Object Type you have selected. This field is the field that you define values for in the left column of the lookup table. To insert entries in the lookup table, right-click in the table grid and choose Insert from the popup menu. This will place a default record in the table. Modify the Field Value that Simulator should look for in the table, and a corresponding Characteristic Value that should be used for the field value given. For example, you could use a lookup table with Line Thickness for transmission line objects, where the thickness of the line would vary based on the amount of flow on the line. Different thicknesses could be assigned to different levels of flow using a lookup table.

Context

In the Context Objects, the user will specify what specific type of graphical objects the dynamic formatting will apply to. This list view will be populated with the Case Information Display object (if the dynamic formatting definitions set correspond to the general case), plus the several graphical objects related to the Object Type. (For buses, for example, it will include the graphical bus, the bus fields, and the bus gauge).

Fields

In the Fields list view, the user will be able to select which fields the dynamic formatting settings will apply to. This view will be populated only for those context objects with fields, such as the Case Information Display and the Object Fields. This list is only visible when you choose certain types of context objects in the context objects list.

Show Only Commonly Used Fields

If this option is checked, only a reduced list of selected fields will be displayed. Otherwise, all the fields belonging to the object will be shown.

Characteristics

The characteristics that the user will be allowed to modify dynamically. These include line thickness, style, color, and background color; font name, size, color, and style; highlight color; surround shape, color and thickness; color and magnification of the 'X' on top of the objects; and blinking color and interval.

Difference Flows

The **Difference Flows** feature provides an easy mechanism for comparing two power system cases. For example, Difference Flows can be used to show the difference in transmission line flows and bus voltages resulting from a contingency or a change in power transfer between two areas. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

The Difference Flows Dialog can be accessed by:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows > Difference Flows** from the [Other Tools Ribbon Group](#).
- Selecting **Difference Flows** from the [online local menu](#).

Use of this feature affects all aspects of the Simulator environment. When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three [Difference Flow Case Types](#) (base, present or difference) is currently being displayed.

All Difference Flows Actions are controlled via the Difference Flows Dialog or the drop-down list on the Difference Flows toolbar button.

Difference Flows: Case Types

When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three case types is currently being displayed. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

The case types available are:

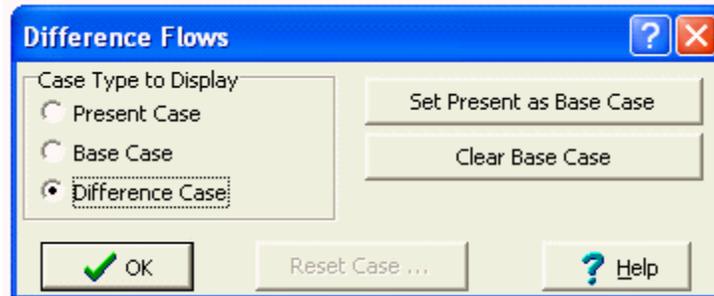
- **Base Case** – A solved power system that serves as the reference for the difference flows tool. To establish a base case, set up a solved power system corresponding to the desired operating point. Open the Difference Flows Dialog and click the button labeled **Set Present as Base Case**.
- **Present Case** – The operating point used in the Difference Flows comparison. **Note:** The Present Case must have the same numbering scheme as the Base Case for proper operation of the Difference Flows tool. See [Topological Differences](#) for more information.
- **Difference Case** – The difference between the Present Case and the Base Case values. The values displayed in the Difference Case are established using the Base Case as the reference. When showing status fields for branches, generators, etc... fields will show up as either OPEN/CLOSED or CLOSED/OPEN if the status changed. The status in the present case will be listed first. A similar format will be used for fields such as generator AGC Status and other text fields.

To toggle between the different case views, open the [Difference Flows Dialog](#) and select the desired case type. Alternatively, you can click the **Difference Flows** button in the [Other Tools Ribbon Group](#) on the [Tools](#) ribbon tab and select the desired case type.

Difference Flows Dialog

To display the Difference Flows dialog:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows > Difference Flows** from the [Other Tools](#) ribbon group. This is available in both [Edit Mode](#) and [Run Mode](#).
- Select **Difference Flows** from the [online local menu](#).



Using Difference Flows

To use the Difference Flows tool:

- Set up a solved power system corresponding to a desired operating point. This operating point will be defined as the [Base Case](#).
- Select **Difference Flows** from the Run Mode ribbon group on the [Tools](#) ribbon tab, or select **Difference Flows** from the [online local menu](#) in Run Mode.
- On the **Difference Flows Dialog**, click the button labeled **Set Present as Base Case**. This stores the current operating point as the Base Case.



Set Present as Base Case

- Define the operating point ([Present Case](#)) for which to perform the difference flows comparison. The Present Case may be developed either by modifying the Base Case as desired and re-solving, or by opening a new case selecting **Open Case** from the [Application Button](#). In the latter situation, the new case that you open must have the same bus numbering scheme as the Base Case.
- See [Difference Flow Case Types](#) for information on toggling between case views. The currently displayed case type is shown in the [PowerWorld Simulator Status Bar](#). When viewing either the Base Case or Difference Case, the corresponding status bar display will be highlighted.



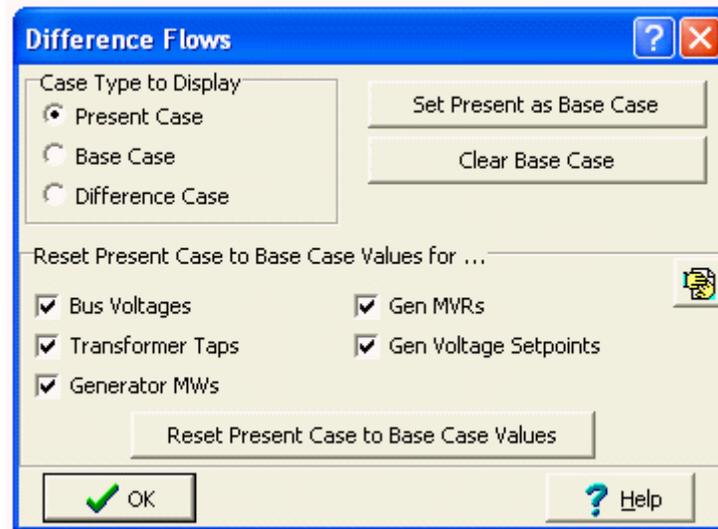
Viewing Difference Case

Note that the Difference Flows tool can only be used in Run Mode and the status bar will not display a case type while in Edit Mode.

When showing the Difference Case, most of the fields shown on the onelines and case information displays show the difference between the present value and its Base Case value. For example, on the Generator Records Display, an entry of 0.0 in the MW field indicates that the real power output of the generator did not change. An entry of 10.0 in the MW field indicates that the present real power output of the generator is 10 MW greater than it was in the Base Case.

At any time during a simulation, you can set the present case as the Base Case by clicking the corresponding button on the Difference Flows Dialog.

Conversely, if you have made changes to the present case, and you wish to revert to some or all of the base case values, you can click on the **Reset Case...** button. When you click this button, the dialog will expand to show you options for resetting specific types of values to their base case values.



Expanded Difference Flows Dialog

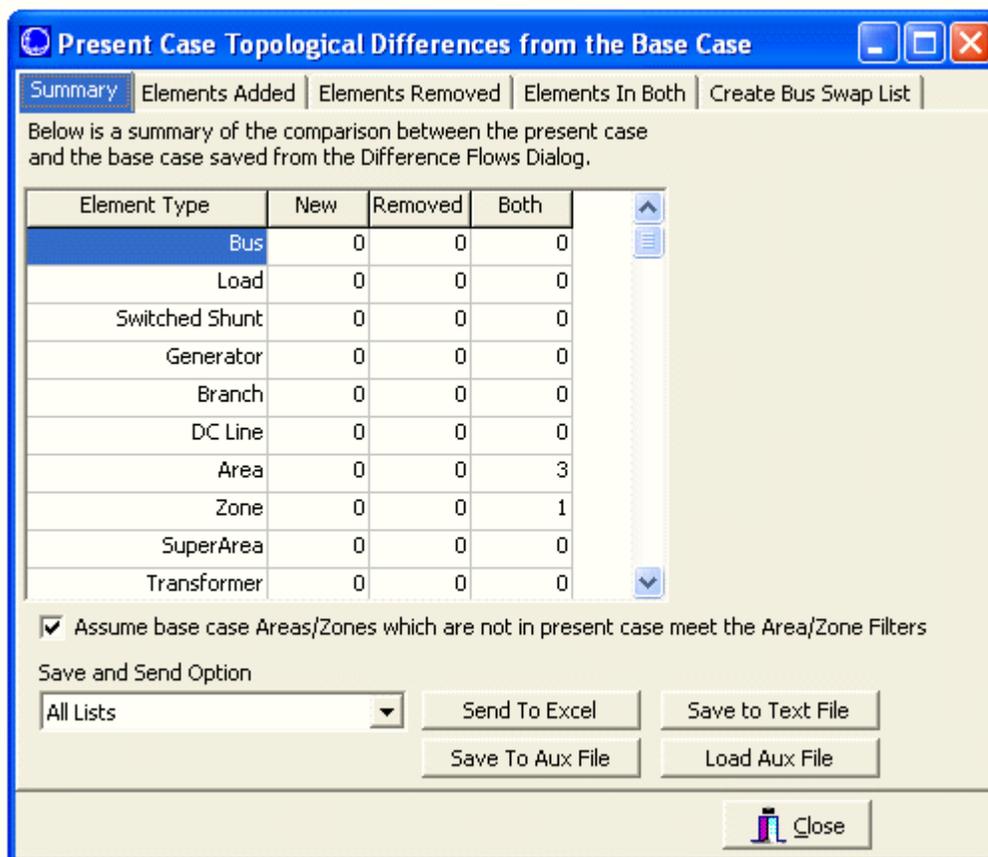
Once you have indicated which types of values you want to reset (by default, all are selected), you can click on the **Reset Present Case to Base Case Values** button to complete the process of reverting to base case values. Note: once you have finished resetting to the base case values, you can hide the Reset options by clicking the Hide Reset Button in the upper right hand corner of the Reset options panel.

The oneline diagrams and case information displays cannot indicate structural differences in the case very well, such as the addition or removal of a device. To identify such differences, make use of the [Present Topological Differences from Base Case](#) tool to identify topology differences.

Present Topological Differences from Base Case

The Present Topological Differences from Base Case option provides the users a way to compare the topological differences between two difference cases in Simulator.

To compare two cases topologically, you must load the first case you wish to use as the reference into Simulator. Use the [Difference Flows](#) tool to set the case as the base case in memory. Once the reference case has been stored as the base case, open the second or comparison case into Simulator. Now if you check the [Tools](#) ribbon tab, and choose the **Difference Flows Menu** on the [Other Tools Ribbon Group](#), you will see the option **Present Topological Differences from Base Case**. Choosing this option will open the dialog to display the topological differences.



Topological Differences Dialog

The Topological Differences Dialog contains five tabs. All but the Summary tab are instances of Case Information Displays and thus exhibit [characteristics and controls](#) similar to other displays.

Summary

A listing of the types of objects in the case. The first column displays the number of NEW devices in the comparison case that were not in the base case. Alternately, the Removed column displays the number of devices in the base case that do NOT exist in the comparison case. The third column simply displays the number of items that were matched between the two cases.

Elements Added

Tabular listings of all objects that exist in the comparison case, but not in the reference case.

Elements Removed

Tabular listings of all objects that exist in the reference case, but are not present in the comparison case.

Elements in Both

Tabular listings of all objects present in both cases.

Create Bus Swap List

Used for setting up a bus renumbering list. It is possible that discrepancies in topology between the comparison case and the reference case can be due to a difference in bus numbering between the two cases. [Renumbering the buses](#) in the comparison case may take care of most of the topology discrepancies that are reported in this instance.

Governor Power Flow

The Governor Power Flow dialog shows all information related to solving a governor power flow. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab. Using this dialog, you can modify settings related to solving the case while on Governor or "Island-Based Automatic Generation Control (AGC)".

The Governor Power Flow Dialog has two tabs: [Options](#) and [Generator Options](#)

Governor Power Flow: Generator Options (Ignore Area/Zone/Owner filter) Tab

This tab sheet presents a case information display with all the generators of the case. The columns shown in the display are helpful when looking at solving a Governor Power Flow.

Governor Power Flow: Options Tab

This section describes the options available on the Options tab of the Governor Power Flow dialog. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab.

This tab sheet presents the following controls:

Disable Automatic Generation Control

Disables automatic MW generation changes during the power flow solution.

Island AGC Tolerance

The ACE mismatch tolerance allowed for the Island to be considered solved in terms of generation dispatch.

Island-Based Automatic Generation Control

Select how the generations should be controlled for the dispatch.

- Disabled: The area and super area dispatch settings from the case will be used
- Use Participation Factors of individual generators: Each generator will have its own participation factor, and will contribute according to its participation factor divided by the sum of all participation factors of all other generators in the same island.
- Calculate Participation Factors from Area Make Up Power Values: Each area is assigned a "factor" as to how it should participate towards the generation dispatch in the island. This is similar to participation factors for generators. The total percentage the area contributes towards the generation change needed in the island is equal to its individual factor divided by the sum of all area make up power factors. Then within each area, the generator participation factors determine how the area's percentage is made up of available generation within the area. The area make up power values can be set in the table of areas on the right hand side of the dialog.
- Dispatch using an Injection Group: The island dispatch will be made up by a combination of generators and loads, defined in an [injection group](#) that can be selected from the dropdown list.

How should reactive power load change as real power load is ramped?

If you are using an injection group with load as part of the dispatch, then you can specify how the reactive power load should respond as the real power demand of loads changes with the dispatch. The reactive power can either be kept at the starting ratio of real and reactive power of the load, or the MVAR amount can change at each load by a specified power factor.

Contingency Analysis: An Introduction

Contingency analysis is a vitally important part of any power system analysis effort. Industry planners and operators must analyze power systems covering scenarios such as the long-term effects on the transmission system of both new generation facilities and projected growth in load. Market analysts and planners must make informed decisions regarding transactions for energy trade - whether that trade is for the next hour or months down the road. PowerWorld Simulator's Contingency Analysis tools provide the ability not only to analyze a power system in its base case topology, but also to analyze the system that results from any statistically likely contingent scenario.

Industry planning and operating criteria often refer to the n-1 rule, which holds that a system must operate in a stable and secure manner following any single transmission or generation outage. In PowerWorld Simulator, the individual contingency conditions can also be tailored to consist of either a single element (such as the loss of a transmission line or transformer), or multiple elements (such as the loss of a generator, several buses and a number of branches simultaneously). See [Available Contingency Actions](#) for a complete list of possible contingency actions.

Simulator can be set to use a Full Newton solution or use a DC Load Flow method to analyze each contingency. The Full Newton approach is not as fast as a DC Load Flow, but the results tend to be significantly more accurate and allow for gauging voltage/var effects.

The [Tutorial](#) is a great place to start learning about using Simulator's Contingency Analysis Tool. We also recommend reviewing the [Terminology](#) used throughout the Contingency Analysis help files prior to continuing.

Available Contingency Actions

The current edition of Simulator can process lists of contingencies including the following:

For Transmission Lines and Transformers (Branches)

- The opening or closing of transmission lines and transformers

For individual Generators, Loads, or Switched Shunts

- Loss or recovery of a particular generator, load, or switched shunt

For all the Generation, Load, or Switched Shunt at a particular bus

- Movement of generation, load, or switched shunt MWs or Mvars to another terminal bus
- Changing or setting of load, switched shunt, or generator MWs or Mvars
- Change or setting of a generator or switched shunt voltage setpoint

For Buses

- Opening of all lines connected to a bus

For Interfaces

- Opening or closing of all lines or transformers in an interface

For Injection Groups

- Opening, closing, or changing of output of all devices in an injection group
- Setting or Changing the net MW Injection by a Percent or MW value and by proportion or by merit order

Series Capacitors

- Bypassing or placing series capacitors in service
- Setting of the Series Reactance by percent or to a particular per unit value

Phase-Shifting Transformers

- Changing or setting the Phase-Shifter Regulation MW value

DC Transmission Lines

- Opening or closing DC Lines
- Changing DC Line setpoint MW or Amp values

Three-Winding Transformers

- Opening or closing all legs of three winding transformers

**Solving the Power Flow
Contingency Block**

Contingency Analysis: Terms

Contingency – Contingencies are the basis of the Contingency Analysis tool. A single contingency can contain a single contingency element (referred to as n-1 contingencies) or multiple contingency elements.

Contingency Action - A statistically likely condition that could occur during power system operation. See [Available Contingency Actions](#) for a listing of actions supported by Simulator.

Model Criteria – Criteria under which a contingency action will occur. For example, the user can specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Simulator allows you to define Model Criteria, which consist of both [Model Conditions](#) and [Model Filters](#). Normally, no Model Criteria will be specified for a given action.

Contingency Element – Consists of a single Contingency Action and its associated Model Criteria, Status and Comment (optional). Multiple Contingency Elements can be defined for a single Contingency.

Contingency Definition – A listing of the Contingency Elements assigned to a Contingency.

Global Action – A [Global Action](#) is a list of contingency elements that occur for ALL contingencies. The elements included in a Global Action do not have to be entered as individual elements in each contingency.

Contingency Block – A list of Contingency Actions. A [Contingency Block](#) can be applied to individual Contingencies by including the block as a Contingency Element.

Contingency Solution Options Dialog

By default, the contingency analysis will use the same options as the power flow algorithm when solving each contingency. You may also override these options for all contingencies, and/or for a specific contingency. This results in the ability to set the power flow solution options in contingency analysis at three different levels

1. Contingency Specific Options (see [Contingency Definition Dialog](#))
2. Contingency Analysis Options (see [Contingency Options Tab](#))
3. General Power Flow Solution Options (see [Power Flow Solution Options](#))

When Simulator executes a particular contingency, it will first look at options specified for that contingency. Any options which are defined for this contingency will be used. Other options set as *Use Default* will look to the Contingency Analysis Options. Again, any options which are defined for contingency analysis will be used. Finally, options marked in the Contingency Analysis Options as *Use Default* will be set to the same setting as the power flow solution options.

In order to specify options for a specific contingency, you click on the **Define/Modify Solution Options** button on the [Contingency Definition Dialog](#). In order to specify options for all contingencies, you click on the **Define/Modify Solution Options** button on the [Contingency Options Tab](#). Both of these will bring up the Contingency Solution Options Dialog.

This dialog contains many options regarding the power flow solution. For options which are a numerical value, just specify a new value to use. For options which are specified by a check-box, the check-box will have three settings: *use option*, *do no use option*, and *use default*. For a more detailed explanation of each option see the [Power Flow Solution Options](#).

To set the values to be the same as the present power flow solution options, click the **Set same as for Power Flow** button. To set all options back to use default, click the **Clear All Settings** button.

Contingency Case References

Contingency Analysis always stores a Reference State or pre-contingency state. The Reference State stores information pertaining to:

- Buses
- Switched Shunts
- Limit Groups
- Loads
- Branches
- Generators
- Areas / Super Areas
- MW Transactions
- Power Flow Solution Options

See [Reference State Information](#) for details on the specific information stored.

The reference state is loaded into memory prior to the execution of each contingency during automatic processing of the contingency list. This ensures that all contingencies start from a common Base Case. Furthermore, the system is set back to the reference state following completion of the automatic processing. Note: The system is not restored to the reference state when the **Solve Selected Contingency** option is selected from the Contingency Record Display's local menu (see [Reference State Solution Options](#) for more information).

The reference state is always stored in Simulator after the first instance of opening the contingency analysis form. Therefore, opening the contingency analysis again may result in a prompt from the program. This prompt will ask you if you wish to set the contingency analysis reference state to the current state of the system (in case you have made changes since the last contingency analysis run), or if you wish to keep the existing contingency analysis reference state (which was set by previously opening the contingency analysis dialog.) Note that if you choose the second option, any changes you may have made to the case outside of the contingency analysis will be lost, as the reference state stored with the contingency analysis tool will reset the system state to the reference state.

Click [here](#) for information on defining the reference state.

Contingency Case References - State Information

Simulator stores the following information with the Contingency Analysis Reference State:

Bus State

- In or out of service
- Voltage Magnitude and angle
- Boolean expression stating whether any load exists at the bus (this is used because some of the contingency actions such as *MOVE GEN* will create a fictitious load if there is no generation at the destination bus)
- MW Marginal Cost

Switched Shunt State

- In or out of service
- Nom value Mvar
- Control mode (FIXED/DISCRETE/CONTINUOUS)
- Nom Value MW
- All Setpoint Values
- Description of Blocks
- Low/high range for voltage control

Limit Group State

- Rating sets for normal operation (Line, Interface...A, B, etc...)

Load State

- In or out of service
- Constant power MW and Mvar components of load
- Constant current MW and Mvar components assuming one per unit voltage
- Constant impedance MW and Mvar components assuming one per unit voltage
- MW Scale
- Mvar Scale
- AGC Status
- Min/Max Load MW

Line State

- In or out of service
- Bypassed?
- Transformer Control?
- Tap ratio
- Phase shift
- High/low desired setpoints
- Series Reactance X (only for series capacitors because a contingency element can change this)

DC Line State (& Multi-Terminal DC Line State)

Generator State

- In or out of service
- MW Output
- Mvar Output
- Max/Min MW Output
- Participation Factor
- Max/Min Mvar Output
- Voltage Setpoint
- Regulated Bus Number
- AGC Status (YES/NO)
- AVR Status (YES/NO)
- Capability Curve
- Use Capability Curve?
- Line Drop Compensation Impedance
- Line Drop Compensation Status (YES/NO/POSTCTG)

Area State and Super Area State

- Unspecified MW Transactions
- MW Scale
- Mvar Scale
- AGC Status
- Use Area Participation Factors? (for Super Area)

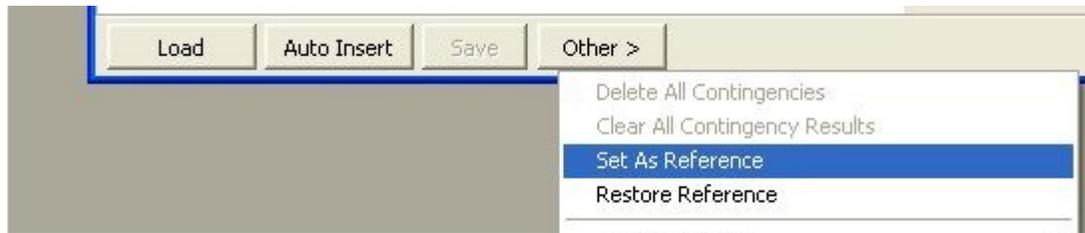
MW Transaction

- MW Value
- Enabled Status

Power Flow Solution Options

Contingency Case References - Defining the Reference State

The reference state is initially defined as the power system state that exists when the contingency analysis is opened for the first time for a given power flow case during a Simulator session. See [Reference State Information](#) for details on the specific information stored in the reference state.

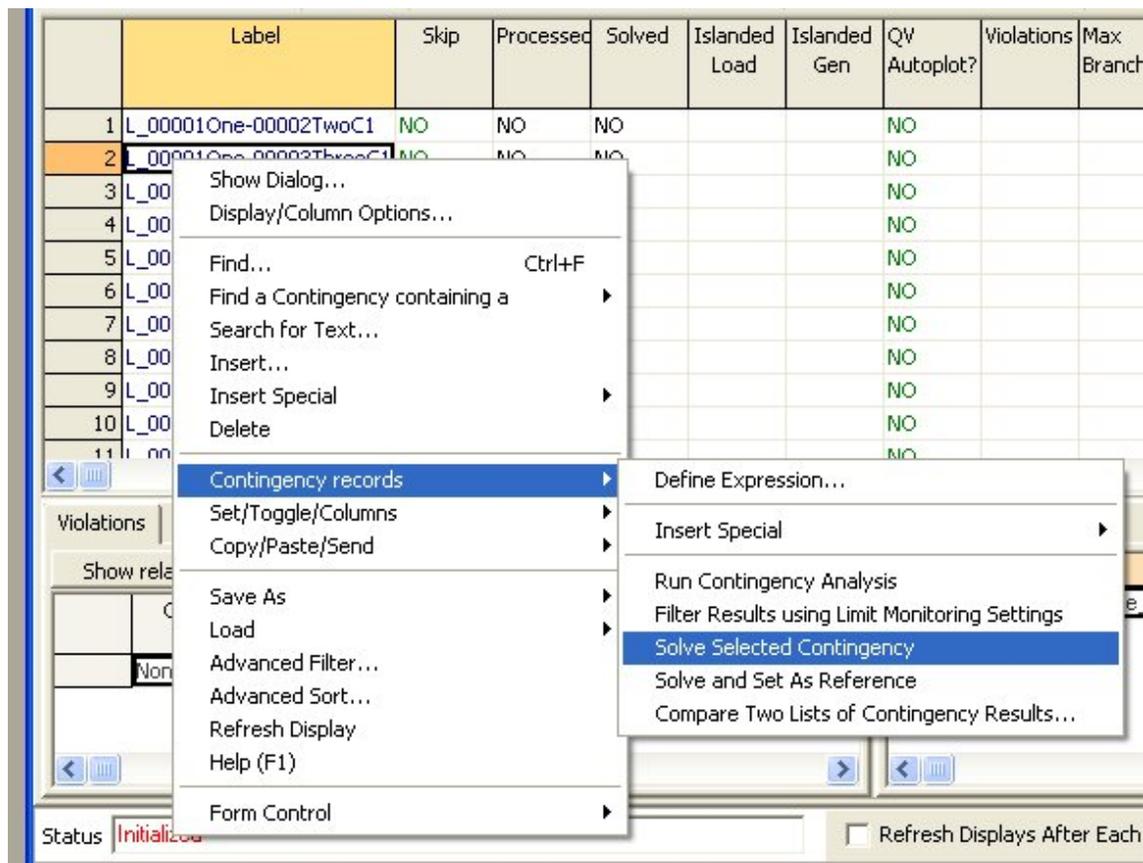


Setting the reference state

The reference state can be changed each time that the Contingency Analysis Dialog is opened after having already established a reference state. The [Setting Reference State When Contingency Analysis is Opened](#) option is available for determining how the reference state should be set when the dialog is re-opened during a Simulator session.

To change the reference while the Contingency Analysis Dialog is open, select the **Set as Reference** option from the [Other >](#) button on the [Contingencies Tab](#) of the Contingency Analysis Dialog. The reference state can also be redefined using the Solve and Set as Reference option from the Contingency Record Display's local menu (See [Reference State Solution Options](#) for more information).

Contingency Case References - Reference State Solution Options


Contingency Record Display Local Menu

When you solve contingencies one at a time, you may choose between the **Solve Selected Contingency** and **Solve and Set As Reference** options from the [Contingency Record Display's local menu](#).

Solve Selected Contingency causes Simulator to first load the reference state into memory then solve the contingency. Following the solution, *the reference state is not restored*; the system state then reflects the power system flows of the post-contingency state. The advantage of this approach is the ability to implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by the contingency. Be aware; however, that prior to solving another contingency, Simulator will reset the system state to reference state thereby removing all modifications made following the previous contingency solution. The user may also automatically restore the system state to reference state by selecting **Other > Restore Reference** from the Contingency Analysis Dialog.

Solve and Set As Reference acts the same as Solve Selected Contingency with one exception. After executing the contingency, the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use the post-contingent state as the Reference State.

Click [here](#) for details on the specific information stored in the reference state.

Click [here](#) for information on defining the reference state.

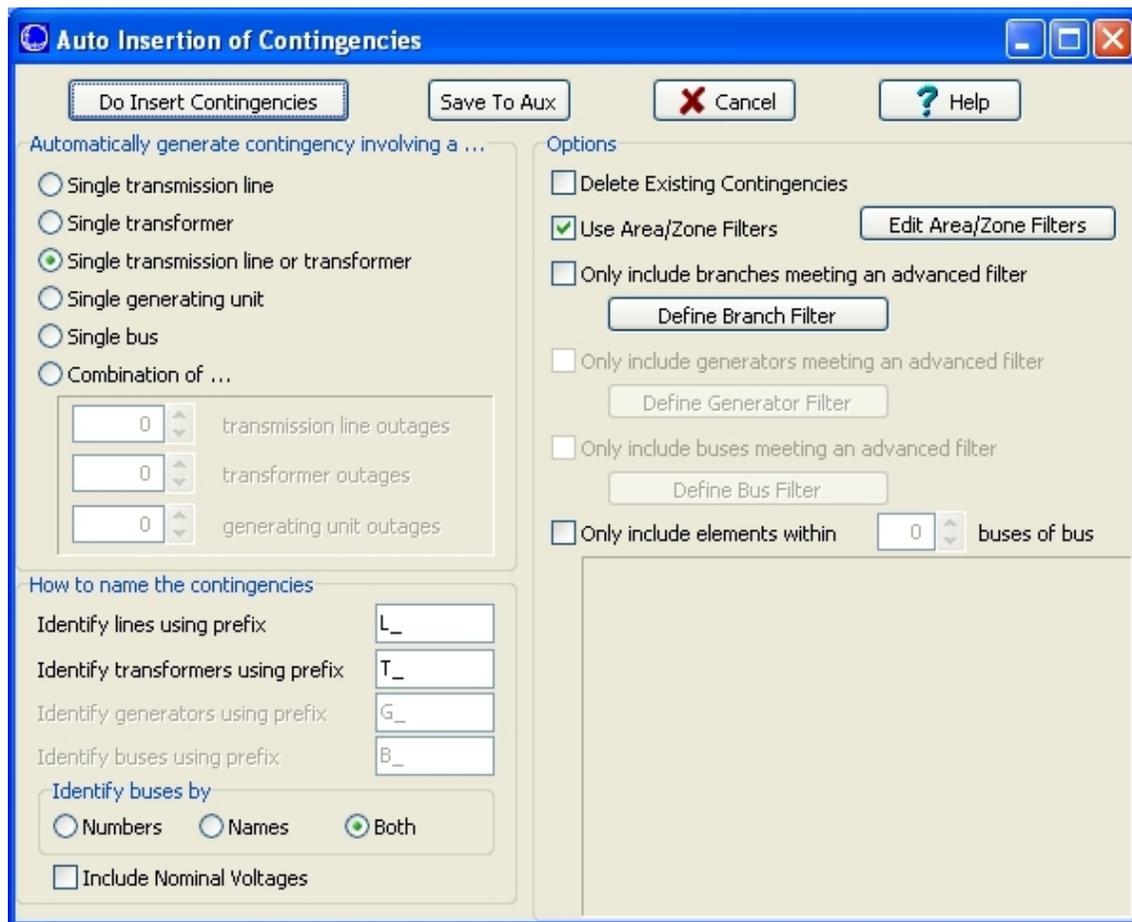
Contingency Analysis: Defining Contingencies

There are four options for defining contingencies. The user may: [Load Contingencies from a File](#), [Auto Insert Contingencies](#), or use the local menu to either **Insert** contingencies or **Quick Insert a Single Element Contingency**.

Auto Insert Contingencies

Simulator allows you to automatically generate a contingency list containing branch, generator and/or bus outages. To accomplish this, click the **Auto Insert** button along the bottom of the [Contingency Analysis Dialog](#) or right-click on the list of contingencies table in the [Contingencies Tab](#) and select **Insert Special > Auto Insert Contingencies....** This opens the **Auto Insertion of Contingencies Dialog** (shown below).

When automatically inserting contingencies, you must specify the type, options and naming conventions you want for the new contingencies. You must also specify whether to delete or retain existing contingencies.



Auto Insertion of Contingencies Dialog

The Auto Insertion of Contingencies Dialog has the following controls:

Do Insert Contingencies

Press the **Do Insert Contingencies** button to generate the contingency list once all other options have been set.

Automatically generate contingency involving a...

The options available in this box define what to add to each automatically inserted contingency element. You can choose single transmission line, transformer, transmission line or transformer, generating unit, or bus contingencies. Choosing one of these options results in each contingency containing only one element of the specific type.

You can also define contingencies containing multiple outages by checking the **Combination of...** option, and then specifying how many of each type of element (Lines, Transformers, and Generating Units) you want considered in the contingency. When you use the **Combination of...** option, Simulator will automatically determine all possible combinations for the element types specified (based on the settings in **Options**) and create the contingencies. Combination contingencies do not currently allow the inclusion of bus contingencies.

Options

When Simulator auto-generates the contingency records for the element types specified, the options in this section will further determine which elements are included and which elements are ignored.

Delete Existing Contingencies

When checked, any previously existing contingency records will be deleted before any automatically created contingencies are inserted.

Use Area/Zone Filters

When checked, the elements included in the auto-generated contingencies will be only elements that are within areas and zones defined in the [Area/Zone/Owner Filters dialog](#). You may edit the Area/Zone/Owner filters by clicking on the **Edit Area/Zone Filters** button.

Only include ... meeting an advanced filter

When checked, the branches, generators, or buses included in the auto-generated contingencies will be only those elements meeting conditions outlined by an [advanced filter](#). To set the conditions to be used, click the respective **Define... Filter** button.

Only include elements within X buses of bus

When checked, only elements that are electrically within X number of buses from the specified bus will be included when creating the contingencies. For example, consider bus 1 is electrically connected to bus 2, which is in turn connected to bus 3. If we specify the bus to be bus 1, and choose to include only elements that are within 0 buses of bus 1, then the contingency record will include the branch between buses 1 and 2, and if desired any generators attached to bus 1. However, the branch between buses 2 and 3 and any devices attached to bus 2 and 3 will NOT be included in the contingency because bus 2 is electrically 1 bus away from bus 1.

To specify the bus used, you can find the bus by using the search engine. The search engine allows you to search by name or number. If you know the bus number, choose Sort by Number, and type the bus number in the search box. If you know the name of the bus, choose Sort by Name, and type the name of the bus in the search box. If you are not sure of the name of the bus, you can use wildcard characters to search through the list of buses until you find the desired bus.

How to name the contingencies

This section allows you to define how each automatically inserted contingency record will be named.

Identify ... using prefix

These four fields allow you to set a specific prefix for generators, lines, transformers and buses so that you can easily determine what type or types of contingencies are modeled in the auto-generated contingency records. By default, the prefixes are G for generators, L for lines, T for transformers and B for buses. However, you can change these prefixes to any character or set of characters you wish.

Identify buses by

This field allows you to specify whether each contingency is labeled using the bus numbers, bus names, or both as identifiers. Whichever type of identifier you choose here will be combined with the defined prefixes to uniquely define the individual contingency elements within each auto-generated contingency record.

Include Nominal Voltages

When this check box is checked, the nominal voltage of buses will be included in contingency labels.

Loading Contingencies from a File

Simulator can load contingency definitions from a text file. The contingencies may be specified in one of three formats:

- [Simulator Auxiliary File Format \(*.aux\)](#) (also see [Contingency Subdata](#))
- Simulator Version 5-7 Contingency File Format (*.ctg) (see the old users manual, or contact PowerWorld Corporation)
- [PTI PSS/E-formatted Contingency Files \(*.con\)](#).

To load contingencies from a text file, click the **Load** button along the bottom of the [contingency analysis dialog](#). A dialog box will be provided for you to specify the file from which to load the contingency records. Specify the file type in the **Files of Type** dropdown box, and select the appropriate file. If contingency records have already been defined for the case with which you are working, you will be asked if you wish to delete the existing contingencies. Respond affirmatively to delete the existing contingencies before adding the new ones from the specified files. Otherwise, click **No**, and the contingencies loaded from the file will be appended to the already existing list.

Simulator is also capable of loading bus throw over data from [PTI PSS/E-formatted load throw over files](#).

PSS/E Contingency Format

Simulator can [read](#) and [write](#) parts of the contingency format used by Power Technology's PSS/E. The current version of Simulator supports all parts of this format, except it does not recognize PTI's Automatic Contingency Specification flags. If you need Simulator to support these keywords, contact PowerWorld Corporation to express your need. Otherwise, we recommend you make use of Simulator's [Auto-Contingency creation tool](#).

PSS/E Load Throw Over Files

Simulator can read [load throw over](#) settings used by Power Technology's PSS/E. These files typically have the extension of *.thr or *.dat. Note, however, that any modifications made to load throw over settings cannot be saved back to the PSS/E throw over file (only read access is currently available).

Saving Contingency Records to a File

Simulator can save contingency definitions to a text file. To save contingencies:

- Click the **Save** button on the [contingency analysis dialog](#). This action will save all contingency definitions and contingency options. Choosing the [Simulator Auxiliary File Format](#) provides a Contingency Settings dialog with options to save related settings such as [limit monitoring settings](#), [general power flow solution options](#), and [list display settings](#) in the same auxiliary file. Contingency Elements may also be defined in the auxiliary file using Bus Numbers, Bus Name and Nominal kV, or [Label](#).
- Select **Save As...** from the [local menu](#) of the [contingency records display](#). Several options for saving contingency information to Simulator's Auxiliary File Format are available:
 - **Auxiliary File...** This action will save the information from the columns presently shown on the contingency records display, plus the contingency elements for each contingency record.
 - **Auxiliary File (only selected records)...** This action will save the information from the columns presently shown on the contingency records display and the contingency records currently selected, plus the contingency elements for each selected contingency record.
 - **Auxiliary File (only selected records/columns)...** This action will save the information from the columns and the contingency records currently selected, plus the contingency elements for each selected contingency record.
 - **Auxiliary File (all contingency related info)...** This action is identical to that of the **Save** button on the [contingency analysis dialog](#) (see above).
 - **Auxiliary File (all contingency related info/only selected records)...** This action is identical to that of the **Save** button on the [contingency analysis dialog](#) (see above), except only definitions of the selected contingencies will be saved.
 - Options for other file formats are also available, as are typically shown on case information [local menus](#). Such other formats are generally not suitable for loading contingency information back into Simulator.
- Click the **Save to Aux** button on the Auto Insertion of Contingencies Dialog. This action will save the Auto Insertion options, but not the contingencies themselves.

The contingencies may be specified in one of three formats

- [Simulator Auxiliary File Format \(*.aux\)](#) (also see [Contingency Subdata](#))
- Simulator Version 5-7 Contingency File Format (*.ctg) (see the old users manual, or contact PowerWorld Corporation)
- [PTI PSS/E-formatted Contingency Files \(*.con\)](#). This format includes an option to truncate the contingency labels to 8 or 12 characters for compatibility with versions of PSS/E that do not support longer labels. The truncated labels will be named such that each is unique. There is also an option to use bus name and bus nominal kV in identification of contingency elements.

To specify the format for the contingency file, set the **Save As Type** option accordingly.

Note that there are limitations when saving to the Simulator Version 5-7 format or the PTI PSS/E formatted files.

Limitations on the Simulator Version 5-7 format

- Does not support the actions SET or CHANGE.
- Does not support the action MOVE, except for Loads.
- Does not support any actions regarding an Interface
- Does not support any actions regarding a Bus

Limitations on the PTI PSS/E format

- Does not support any actions regarding an Interface

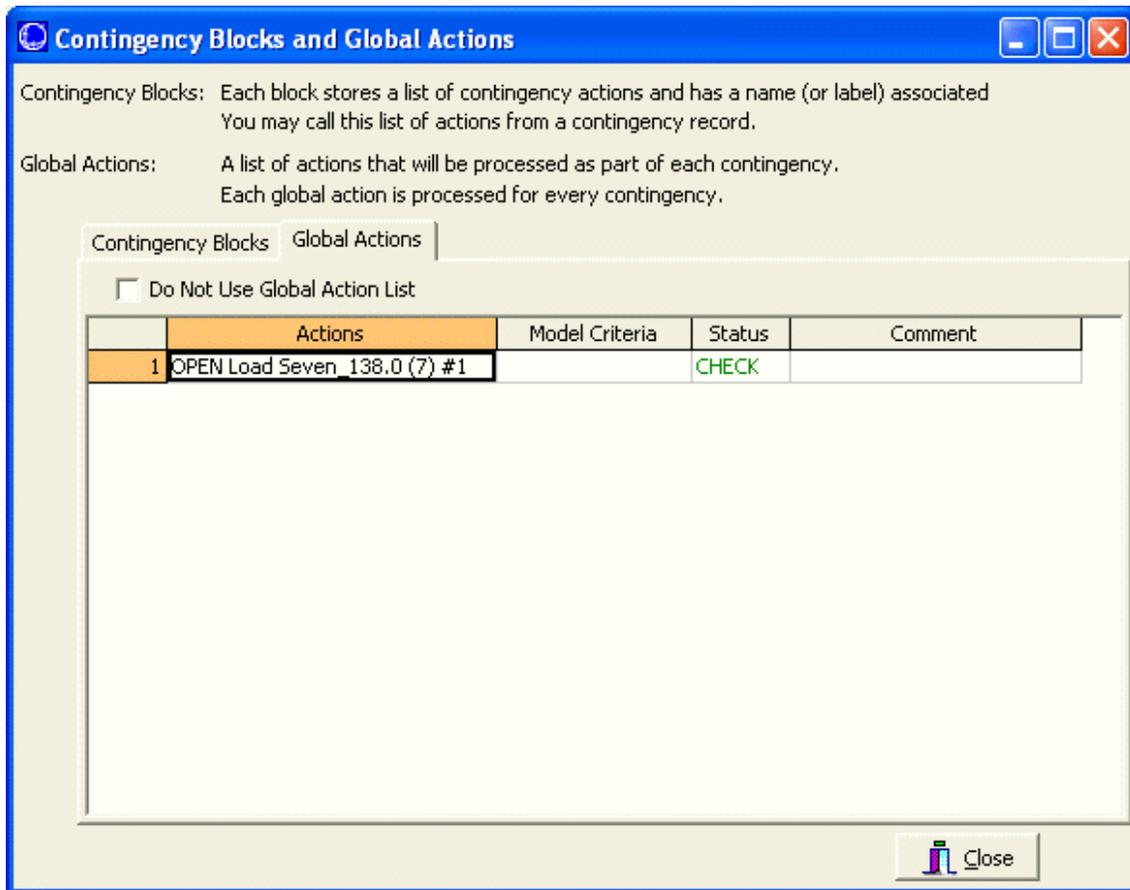
Global Actions

Global actions allow you to define a list of contingency elements that occur for ALL contingencies and do not have to be entered as individual elements in each contingency. These contingency elements are defined and processed in the same manner as contingency elements defined with specific contingencies.

Global actions are defined by clicking the **Contingency Blocks and Global Actions** button on the [Advanced Modeling](#) tab of the [Options](#) page of the Contingency Analysis dialog. Click on the **Global Actions** tab of the Contingency Blocks and Global Actions dialog to define Global Actions. Adding global actions occurs in a manner identical to that of the [Contingency Definition Dialog](#). Right-click in the Actions list to open the [Contingency Element Dialog](#), which allows you to add contingency elements to be treated as Global actions.

The format of the string that describes the actions and the **Model Criteria** and **Status** fields are set in the same manner as described in the [Contingency Definition Display](#).

To disable the use of the Global Actions, check the **Do Not Use Global Action List** option.



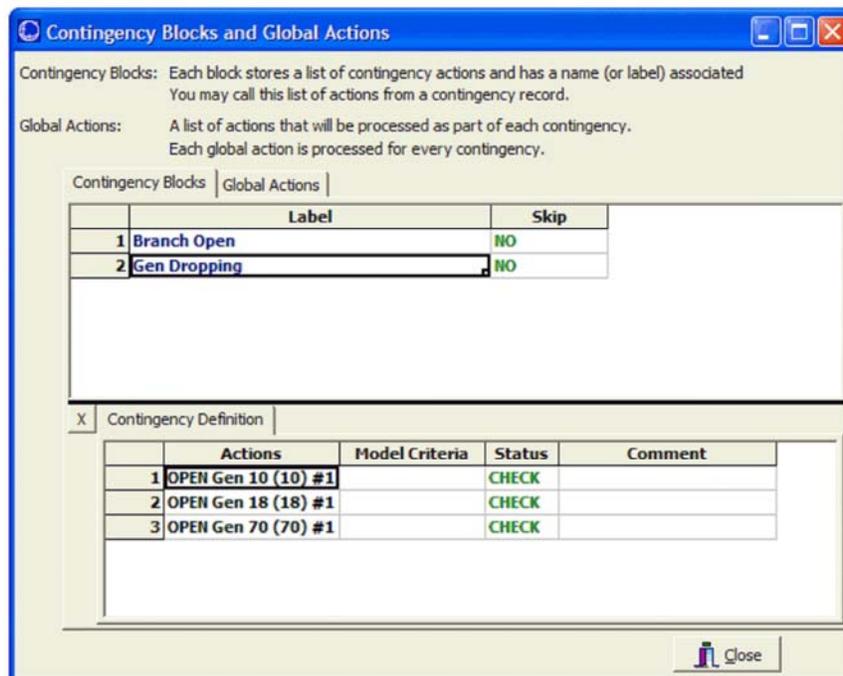
Contingency Blocks

Contingency Blocks are very similar to a [Contingency Record](#), however no results can be associated with them. Contingency Blocks consist of a list of contingency actions. The block is then given a name so that any Contingency Record can call on a Contingency Block. When a contingency block is included as part of a contingency, the Contingency Record will incorporate all the actions from the contingency block into the actions performed by the contingency.

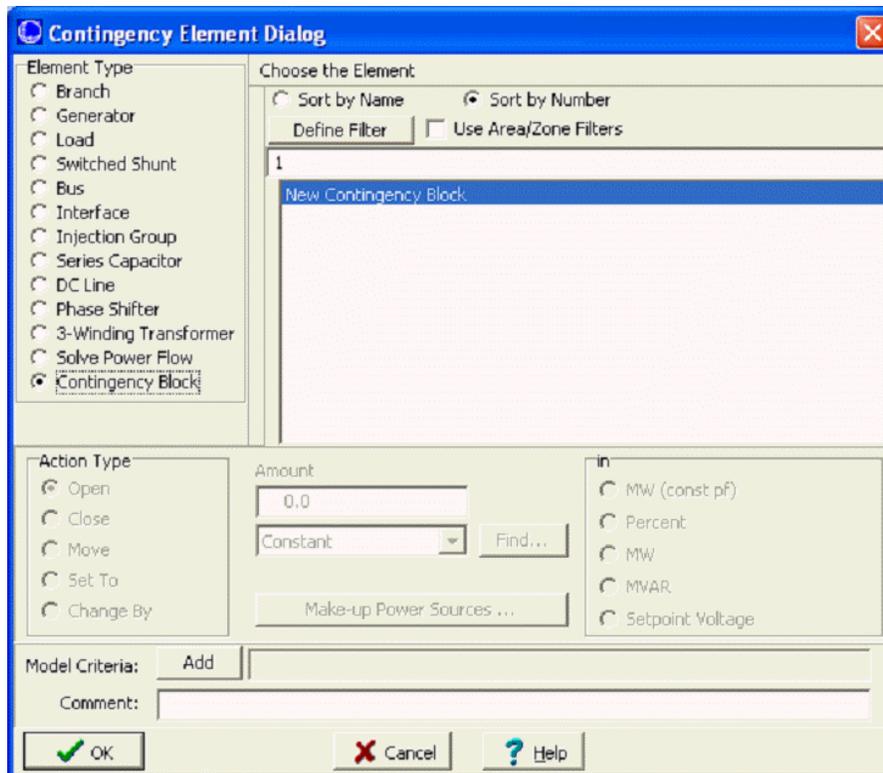
Contingency blocks are defined by clicking the **Contingency Blocks and Global Actions** button on [Advanced Modeling tab](#) on the Options page of the Contingency Analysis Dialog. Click on the Contingency Blocks tab of the Contingency Blocks and Global Actions dialog to show a display for defining Contingency Blocks. Right-clicking in the Contingency Blocks grid and choosing **Insert** allows you to insert a new Contingency Block. A dialog very similar to the [Contingency Definition Dialog](#) will open. Use this dialog to insert elements into the contingency block, create additional contingency blocks, or modify existing contingency blocks. Once a new contingency block has been created, actions can be added to the block by right-clicking in the Contingency Definition grid and choosing **Insert**. This will open the [Contingency Element Dialog](#) that allows the addition and modification of elements in the contingency block. When using the Contingency Element Dialog with Contingency Blocks, the option to insert an Element Type of *Contingency Block* is not available.

The format of the string that describes the contingency block actions and the **Model Criteria** and **Status** fields are set in the same manner as described in the [Contingency Definition Display](#).

To disable the use of a Contingency Block, set the **Skip** field to *YES*. When a Contingency Block is disabled, all of the actions defined in the block will be ignored in all of the contingencies in which the block is included.



Once Contingency Blocks have been defined, a new Element Type, *Contingency Block* will appear on the [Contingency Element Dialog](#). This is shown in the image below. You may then add a contingency block to a contingency by right-clicking on a contingency in the [Contingency Records](#) list display, choose show dialog (or insert if you are adding a new contingency record), click on Insert New Element, and choose Contingency Block from the list of element types.



Using a contingency block is an easy way to include a set of common actions in multiple contingency scenarios, without having to re-define the actions for each contingency.

Contingency Analysis Dialog Overview

Simulator's contingency analysis tools can be accessed only from [Run Mode](#). To access this dialog go to the [Tools](#) ribbon tab, and choose **Contingency Analysis** from the [Run Mode](#) ribbon group. This will open the Contingency Analysis Dialog.

Each tab of the contingency analysis dialog covers a particular aspect of the analysis.

The [Contingencies Tab](#) is used primarily to manage the contingency list and to learn basic information regarding each contingency and the violations that it causes.

The [Lines, Buses, Interfaces Tab](#) is similar to the Contingencies Tab. The Contingencies Tab lists each contingency and shows which power system element violations occur under each contingency. The Lines, Buses, Interfaces Tab lists each power system element and shows which contingencies cause violations on those elements.

The [Options Tab](#) enables you to dictate various parameters for the analysis that govern such things as how violations are flagged in both the Base Case and for contingency conditions and what information should be included in the contingency report.

The [Summary Tab](#) chronicles the performance of the contingency analysis

The bottom panel of the dialog houses a row of buttons whose function at any given time depends on which of the four tabs is visible. The bottom panel also houses a text status indicator to keep you abreast of the current state of the analysis.

Running the Contingency Analysis

To run the contingency analysis means to model and solve one or more contingencies from the case's current contingency list. Simulator's [Contingency Analysis Dialog](#) gives you several options for running the contingency analysis. You may

- Run every contingency in the contingency list (except, of course, for those you have designated to skip using the [Contingency Records Display](#)).
- Run a selected contingency to identify its limit violations and then leave the system in this post-contingency state. (Realize that prior to solving another contingency, Simulator will reset the system state to the reference state. For more information see [Contingency Case References](#)).
- Run a selected contingency to identify its limit violations and keep the resulting case as the new reference point for further contingency analysis runs.

To run the complete contingency list, do any one of the following:

- Right-click on the [Contingency Records Display](#) to bring up its [local menu](#) and select **Run Contingency Analysis**.
- Click the **Start Run** button visible from the [Contingency Tab](#).
- Click the **Start** button visible from the [Summary Tab](#).

To pause the contingency run once it has started, click either **Pause Run** on the [Contingency Tab](#) or **Pause** on the [Summary Tab](#). To resume a paused contingency run, click the **Continue** buttons on either the [Contingency Tab](#) or the [Summary Tab](#). Finally, to terminate a contingency run, click the **Abort** button on the [Summary Tab](#). The status indicator will inform you of the run's current state.

To solve a single contingency, identify its violations, and then leave the system in this post-contingency, select the contingency you wish to model in the [Contingency Records Display](#), right-click to invoke the display's [local menu](#), and select **Solve Selected Contingency**. (Realize that prior to solving another contingency, Simulator will reset the system state to the reference state).

To solve a single contingency and set it as the reference (starting) case for further contingency analysis activity, select the contingency you wish to model in the [Contingency Records Display](#), right-click to invoke the display's [local menu](#), and select **Solve and Set As Reference**. Simulator will model the selected contingency, flag its violations, and leave the resulting case in memory so your future work will affect the post-contingency system rather than the original pre-contingency state. See [Contingency Case References](#) for more information.

Other Contingency Actions

By clicking the **Other** > button on either the [Contingency Tab](#) or the [Lines, Buses, Interfaces Tab](#) of the [Contingency Analysis Dialog](#), you have access to several actions.

Delete All Contingencies

Click this to delete all the contingencies presently stored in memory.

Clear All Contingency Results

Click this to clear all the results of the contingencies from memory. This will not delete the contingencies.

Set As Reference

Click this to set the case presently in memory as the Reference State for contingency analysis. For more information on the reference state, see [Contingency Case References](#).

Restore Reference

Click this to set the state of the present power system case back to the contingency analysis reference state. For more information on the reference state, see [Contingency Case References](#).

Combined Tables

Contingency Violation Matrices

Clicking **Process Contingency Results** will go through the existing contingency analysis tool results and build tables showing the limit violation values for all violated lines, interfaces, and buses and the contingencies for which each element is violated. Upon opening, it is possible to limit processing to branches, buses, and/or interfaces. Also, you can specify whether to display percentages or actual flows in the resulting tables.

Contingency Violation List

Contingency Violation List lists all violations from the current contingency set in a tabular format. This lists eliminates possible clutter by not listing those contingencies for which there are no violations.

Contingency Element Definitions

Contingency Violation List lists all contingency actions from the current contingency set in a tabular format.

Produce Report

To produce a detailed report of the results of the contingency analysis, click **Produce Report**. This will launch a save window that will save the information you customized on the [Contingency Analysis Options: Report Writing](#) page. You will also be given the option of viewing the report in WordPad immediately after creating the file.

What Actually Occurred?

After running contingency analysis, click this to produce a list of actions that occurred during the selected contingency. This is especially useful for identifying those actions conditional on Model Criteria were applied.

Compare Two Lists of Contingency Results

Click this to open a dialog which allows you to specify two sets of contingency analysis results to compare. For more information on this comparison, see [Comparing Contingency Analysis Results](#).

Filter Results Using Limit Monitoring Settings

Click this to filter the contingency analysis results using the present [Limit Monitoring Settings](#). This action will go through each violation for each contingency and verify that the element that was violated is set to be monitored. If the element is not set to be monitored, then Simulator will flag the violation internally as **inactive**. You will then not be able to see these violations on any of the displays, although they will still be saved in memory until you delete these contingencies, or reprocess them. These violations will also not be saved when you choose to save the contingency results.

Because the violations are saved in memory however, you can get them back without reprocessing the contingency list. To do this, change your Limit Monitoring Settings so that those violations will be set for monitoring again. Then click this option again.

When using this option, keep in mind that Simulator cannot filter results that do not exist in memory. If the filter is to be used to filter out different percentages of overload, then the contingency analysis should be run with the lowest percentage desired. For example, if the filter option is to be used to give all of those elements that are loaded above 70% of their limit and then those that are loaded above 90% of their limit, the contingency analysis should be run with the Limit Monitoring Settings defined to monitor elements that are loaded at 70% of their limit. Also, keep in mind that the more elements are monitored the slower the processing will be.

Auto-fill Blank Contingency Element Comments

Selecting this option will fill the Comment field of the Contingency actions in the [Contingency Definition](#) table with a copy of the contingency action definition. The action description itself cannot be modified, but the comment can be modified to be more descriptive of the action being taken, for your own reference.

Sensitivity Calculations >

Simulator provides the [PTDF tool](#) for calculating the impact of a MW transfer on all the transmission lines in the system. The Simulator [ATC tool](#) further extends the linearized methods by integrating linearized contingency analysis with the PTDF calculations. The sensitivity calculations provided here are an extension of this. They allow you to ask the question, **How will each contingency-caused branch or interface violation be affected by a MW transfer?**. Note: this calculation is not relevant for bus violations.

Calculate OTDFs using existing PTDFs

Before executing this, you must first go to the [PTDF Dialog](#) and calculate the PTDFs for the transfer direction you are interested in. These PTDF values will then be used throughout the OTDF calculation. Click this to calculate OTDFs for each contingency-caused branch or interface violation. The values calculated will be a measure of what percent of a transfer would appear on the branch or interface after the respective contingency occurs. Realize for branch violations, that the sign of the OTDF value will be relative to the direction of the MW flow found during the contingency analysis (see the Element description on the [Contingency Violations Display](#)).

Filter out Violations Using OTDFs

Once you have calculated the OTDFs using the existing PTDFs, you can then filter the results by selecting this option. A dialog will appear for you to enter a minimum OTDF value. All violations that have an OTDF smaller than this number will be flagged as **inactive** and will not show up in the list of violations. See the note above regarding the **Filter Results using Limit Monitoring Settings** to better understand how inactive violations are treated.

Contingency Analysis Dialog - Contingencies Tab

The **Contingencies Tab** of the [Contingency Analysis Dialog](#) provides tools for managing and simulating lists of contingencies. The top portion of the Contingencies Tab lists the contingency records that have been defined for the case. This table is called the Contingency Records Display. The contingency records display is a type of [case information display](#) and thus shares many of the properties and controls common to all other case information displays. By right-clicking on the display, you gain access to its [local menu](#), which offers several choices:

- **Insert** allows you to insert a new contingency record
- **Insert Special** gives the following options:
 - **Quick Insert of Single Element Contingency** which allows you to quickly specify a single element contingency via the [Contingency Element Dialog](#)
 - **Auto Insert Contingencies** which opens up the [Auto Insert Contingencies](#) dialog
 - **Merge Pairs of Selected Contingencies** which generates every possible pair of contingencies based on the selected contingencies. For instance, if you select 4 contingencies (A, B, C, D), then click **Merge Pairs of Selected Contingencies**, 6 new contingencies will be generated: A+B, A+C, A+D, B+C, B+D, and C+D.
 - **Clone Contingencies** which makes copies of each selected contingency
- **Delete** allows you to delete a particular contingency
- **Show Dialog** displays the [Contingency Definition Dialog](#) corresponding to a particular contingency
- **Contingency Records** gives the following options:
 - **Run Contingency Analysis** (see [Running the Contingency Analysis](#))
 - **Filter Results Using Limit Monitoring Settings** (see [Other Contingency Actions](#))
 - **Solve Selected Contingency** (see [Contingency Case References - Reference State Solution Options](#))
 - **Solve and Set as Reference** (see [Contingency Case References - Reference State Solution Options](#))
 - **Compare Two Lists of Contingency Results** (see [Other Contingency Actions](#))
 - **Apply Selected Contingency** which applies the actions in the contingency definition, but does not solve the contingency or calculate violations.
 - **What Actually Occurred?** (see [Other Contingency Actions](#))
- Many other options (e.g., printing, finding, and sorting) which are characteristic of [case information displays](#).
- **Save As** gives several options characteristic of case information displays as well as options unique to the contingency records display:
 - **Auxiliary File (all contingency related info)** first opens a file dialog for specification of the auxiliary file. Clicking Save from the file dialog opens a **Contingency Settings** dialog for specification of related settings to save in the same auxiliary file, including [Contingency Options](#), [Limit Monitoring Settings](#), [General Power Flow Solution Options](#), List Display Settings ([Case Information Customization](#)), and Contingency Results. Definitions of all contingencies are also saved. This has the same functionality as clicking the **Save** button in the bottom panel.
 - **Auxiliary File (all contingency related/only selected records)** has the same functionality as **Auxiliary File (all contingency related info)**, except contingency definition(s) are only saved for the selected contingency record(s). You may also specify related settings to save.

You can also sort the display's contents by any field just by clicking on the field's heading. The default fields shown in the contingency records display are described at the bottom of this page.

As you scroll through the records in the contingency records display, you will notice that the contents of the tables that occupy the middle third of the contingency analysis dialog change. These tables are the [Contingency Definition Display](#) and the [Contingency Violations Display](#). These displays show the violations and definition for the contingency that is selected in the contingency records display at the top of the dialog. You may optionally hide the Contingency Definition Display by clicking on the X to the upper right of this display. To show the display again, click on the O to reopen it. You may change the relative width of the Contingency Definition and Violations display by moving your mouse over the line between the displays until your cursor changes. Then left click and drag to modify these widths.

When you first load a new contingency list into memory, the current status indicator along the bottom of the display will indicate the contingencies have been Initialized. During a contingency analysis run, the current status indicator may take on the values *Running*, *Paused*, *Aborted*, or *Finished*.

The contingency tab of the contingency analysis dialog offers several ways to run the contingency analysis. To start a run, you may click the **Start Run** button. Alternatively, you may choose **Run Contingency Analysis** from the [local menu](#) of the contingency records display. Once a contingency analysis run has started, you may pause it at any time by clicking the **Pause Run** button, after which you may resume the run by clicking **Continue**. In addition to running the full set of contingencies, you may also choose to run just a single contingency. See [Running the Contingency Analysis](#) for more details.

Several other actions related to contingency analysis are also available from the Contingencies Tab. These are accessed by clicking on the **Other Actions >** button. They are described on the [Other Contingency Actions](#) page.

You may close the Contingency Analysis Dialog at any time either by clicking **Close** or by selecting **Close** from the [local menu](#) of the Contingency Records Display or the [Contingency Violations Display](#).

By default, the contingency records display presents the following fields:

Label

The name of the contingency.

Skip

Indicates whether Simulator should skip the corresponding contingency in performing the contingency analysis. If the value of the Skip field is *Yes* for a contingency, then that contingency will not be implemented when performing the contingency analysis. This is a toggleable field, which means that you can toggle its value by double-clicking the field.

Processed

Indicates whether the contingency has been analyzed yet as part of the current contingency run.

Solved

If the contingency has not yet been processed, which means that the contingency has not yet been implemented, then the value of this field is *No*. For contingencies that have been processed, the Solved field indicates whether the power flow case that resulted from the contingency could be solved to within tolerance. If the resulting power flow case could not be solved, you should investigate the contingency closely to determine if that contingency is indeed harmful to the stability of the system.

Post-CTG AUX

The auxiliary file specified here will be loaded at the start of this contingency, which can be used to alter the reference state for this contingency. If an auxiliary file is specified here, then the contingency options [Advanced Modeling](#) Post-Contingency Auxiliary File will not be loaded for this contingency

Double-click a cell to edit the specified post-contingency auxiliary file. A specified post-contingency auxiliary file may be removed using the [Contingency Definition Dialog](#).

Note: Only data stored with the contingency reference state will be "reset" when the reference state is restored. Therefore, only data stored with the reference state should be loaded via a post-contingency auxiliary file. Click [here](#) for details on the specific information stored with the reference state.

Islanded Load

Displays the sum of the amount of load that was islanded from the rest of the system due to the contingency. This load is inaccessible from the rest of the system.

QV Autoplot?

This option must be toggled to *Yes* if you intend to run a [QV Analysis](#) and automatically plot the QV curve for one or more contingency scenarios.

Violations

Identifies the number of violations caused by the particular contingency. This number represents the total number of contingencies (branch thermal violations + bus violations + interface violations) that were caused by the contingency. Depending on how you have configured the reporting of Base Case violations (see Base Case of the Options Tab), this number may include all, some, or none of the violations that were present in the Base Case model.

Max Branch %

Indicates the percentage overload of the worst-case branch violation. If there are no branch violations, this field will be blank.

Min Volt

Indicates the lowest bus voltage resulting from the contingency. If there are no low voltage violations, this field will be blank.

Max Volt

Indicates the highest bus voltage resulting from the contingency. If there are no high voltage violations, this field will be blank.

Max Interface %

Indicates the percentage overload of the worst-case interface violation. If there are no interface violations, this field will be blank.

Contingency Violations Display

The Contingency Violations Display is used to list the violations that were caused by the contingency selected in the [Contingency Records Display](#).

The contingency violations display lists all of the power system elements that become violated as a result of the selected contingency. If you have selected a violation, you may click on the **Show related contingencies** button to view all contingencies that cause a violation on this power system element. Clicking this button automatically moves you to the [Lines, Buses Interface tab](#), and selects the appropriate power system element.

If the contingency selected in the Contingency Records Display resulted in no violations or has not yet been processed, the Contingency Violations Display will display the words None Defined. This display is a type of [Case Information Display](#) and thus shares many characteristics and controls common to all other case information displays. You can sort the list of violations by any field simply by clicking on that field's caption.

The **Combined Tables** button provides access to the Combined Tables options described in [Other Contingency Actions](#). In addition to these options, there is one other option for displaying **What actually occurred?** Selecting this will display a dialog box giving the details of the actions applied during the contingency and any actions that may have been skipped.

By default, the Contingency Violations Display contains the following fields:

Category

The type of violation that occurred. If the violation is only due to the options set in [Advanced Limit Monitoring](#), then the type will start with the word *Change*.

Element

A character string that describes the element that suffered the violation. This can either be a branch, a bus, or an interface. When the element is a branch, this string provides you with three pieces of information:

- The branch that was violated
- The terminal of the branch which had the highest loading
- The direction of the MW flow on this branch

Example 1: Jamie (22) -> Amy (33) CKT 1 at Amy (33)

This means that a branch connecting Jamie(22) to Amy (33) with circuit ID 1 is violated. The violation is at the Amy(33) terminal. The -> indicates that the MW flow on this line is from Jamie toward Amy.

Example 2: Xena (55) <- Harley (77) CKT 1 at Harley (77)

This means that a branch connecting Xena (55) to Harley (77) with circuit ID 1 is violated. The violation is at the Harley (77) terminal. The MW flow on this line is from Harley toward Xena.

Value

Indicates the value of the violating quantity. For example, if the category of the violation is *Branch Amp* and the Value field is x, then the current on the violated element is x.

Limit

Identifies the limit value that was violated. For example, if the category of the violation is *Branch Amp* and the Limit field is y, then the limit on the current that may flow through the element is y.

Percent

The actual flow value for the element as a percentage of the limit.

Area Name Assoc.

Lists the areas with which the violated element is associated. If the element is a branch, Area Name identifies the area in which each of the branch's terminal resides. If the element is a bus, Area Name identifies the area in which the bus resides. If the element is an area-to-area interface, Area Name will identify the areas that the interface ties; otherwise, it will read *N/A*.

Nom kV Assoc.

Identifies the maximum voltage level associated with the violated element. If the violated element is a branch, then Nom kV lists the nominal voltage of its higher-voltage terminal. If the violated element is a bus, then Nom kV simply identifies the bus' nominal voltage. If the violated element is an interface that is made up strictly of branches, Nom kV lists the maximum nominal voltage of its terminals; otherwise, it will appear as *-9999.9*.

Contingency Definition Display

The Contingency Definition Display lists the [Elements](#) assigned to the selected [Contingency](#). This display appears on both the [Contingency Tab](#) and the Lines, Buses, Interfaces Tab of the [Contingency Analysis Dialog](#), it also appears on the [Contingency Definition Dialog](#).

Select **Insert** from the local menu to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency.

The contingency definition display is a type of [Case Information Display](#) and shares many characteristics and controls common to all other case information displays.

The Contingency Definition Display always contains the following fields:

Actions

This shows a string which describes the action. You may customize the format of the string that describes the contingency actions by right-clicking on the Contingency Definition Display and choosing **Display Descriptions By**, and then choosing *Name*, *Num*, *Name/Num*, *PW File Format by Numbers*, *PW File Format by Name/kV* or *PTI File Format*.

Model Criteria

Simulator allows you to define Model Criteria, which can consist of both [Model Conditions](#) and [Model Filters](#). These specify a criterion under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the **Model Condition and Filter option** on the [Advanced Modeling tab of the Contingency Options](#) page. Model Criteria can be defined from the local menu (**Define Model Criteria** option) on the Contingency Definition Display, from the **Add** (Model Criteria) button on the [Contingency Element Dialog](#), or by using the **Model Conditions** and **Model Filters** buttons on the [Advanced Modeling tab of the Contingency Options](#) page.

Status

Double click the Status field to toggle through the available options. The possibilities are:

- **Check** : The action will only be executed if the Model Criteria are true or if no Model Criteria are specified. **Check** is the default status setting.
- **Always**: The action will always be executed, regardless of the Model Criteria.
- **Never**: The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- **PostCheck**: This action will be considered AFTER all **Check** and **Always** actions have been performed and the load flow solution solved. If the Model Criteria specified for the **PostCheck** action are met in the solved load flow solution (or if no Model Criteria are specified), then this action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped. This process repeats recursively for all Postcheck actions until complete. This behavior only occurs when the [Calculation Method](#) for the analysis is set to *Full Power Flow*. Because the power flow is not solved when either of the two dc methods is selected, the **PostCheck** status instead acts as a **Check** status.

Comment

An optional user-specified data string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under

the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#).

Lines, Buses and Interfaces Tab

The Lines, Buses and Interfaces Tab contains four sub-tabs: Lines/Transformers, Buses, Interfaces, and Nomogram Interfaces. The information contained on each of the sub-tabs provides an alternate method of viewing information similar to that contained on the [Contingencies Tab](#). The individual tabbed sheets show all model objects defined in the case (subject to area/zone/owner and advanced filters) whether each is associated with a specific contingency or not. The user can select any model object on its respective sheet to see how many times a violation occurred on the device during a run of a set of contingencies. When a particular device is selected that had at least one violation during the contingency run, the two pages at the bottom give the details of the analysis for the selected device.

Note: the information contained in the [Contingencies](#) and [Contingency Definition](#) sections of the Lines, Buses and Interfaces tab is object specific. The information present only pertains to contingencies that resulted in violations on the selected object.

Lines, Buses and Interfaces Tab - Contingencies Section

The Contingencies section gives a list of all the contingencies that caused a violation on the selected device during the analysis. This display is very similar to the [Contingency Violations Display](#). While the Contingency Violations Display shows the *elements violated under the contingency*, this display shows the *contingencies that caused the violation*. If you then select one of the contingencies in this list, the [Contingency Definition section](#) displays the actions that took place during the selected contingency. Also, when you have selected a contingency from this list, you can click the **Show Other Violations** button. This will change your dialog to the contingencies tab and select the contingency you have selected, thus allow you to see other violations caused by this contingency.

The **Combined Tables** button provides access to the Combined Tables options described in [Other Contingency Actions](#). In addition to these options, there is one other option for displaying **What actually occurred?** Selecting this will display a dialog box giving the details of the actions applied during the contingency and any actions that may have been skipped.

This page provides an easier way to check the contingency results when you are concerned with the results for a specific device in the system. The Lines, Buses and Interfaces Tab provides a much easier tool for this kind of examination, as opposed to looking through each contingency on the Contingencies Tab and trying to find each instance of a violation on the desired element.

Contingency Definition Display

The Contingency Definition Display lists the [Elements](#) assigned to the selected [Contingency](#). This display appears on both the [Contingency Tab](#) and the Lines, Buses, Interfaces Tab of the [Contingency Analysis Dialog](#), it also appears on the [Contingency Definition Dialog](#).

Select **Insert** from the local menu to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency.

The contingency definition display is a type of [Case Information Display](#) and shares many characteristics and controls common to all other case information displays.

The Contingency Definition Display always contains the following fields:

Actions

This shows a string which describes the action. You may customize the format of the string that describes the contingency actions by right-clicking on the Contingency Definition Display and choosing **Display Descriptions By**, and then choosing *Name*, *Num*, *Name/Num*, *PW File Format by Numbers*, *PW File Format by Name/kV* or *PTI File Format*.

Model Criteria

Simulator allows you to define Model Criteria, which can consist of both [Model Conditions](#) and [Model Filters](#). These specify a criterion under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the **Model Condition and Filter option** on the [Advanced Modeling tab of the Contingency Options](#) page. Model Criteria can be defined from the local menu (**Define Model Criteria** option) on the Contingency Definition Display, from the **Add** (Model Criteria) button on the [Contingency Element Dialog](#), or by using the **Model Conditions** and **Model Filters** buttons on the [Advanced Modeling tab of the Contingency Options](#) page.

Status

Double click the Status field to toggle through the available options. The possibilities are:

- **Check** : The action will only be executed if the Model Criteria are true or if no Model Criteria are specified. **Check** is the default status setting.
- **Always**: The action will always be executed, regardless of the Model Criteria.
- **Never**: The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- **PostCheck**: This action will be considered AFTER all **Check** and **Always** actions have been performed and the load flow solution solved. If the Model Criteria specified for the **PostCheck** action are met in the solved load flow solution (or if no Model Criteria are specified), then this action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped. This process repeats recursively for all Postcheck actions until complete. This behavior only occurs when the [Calculation Method](#) for the analysis is set to *Full Power Flow*. Because the power flow is not solved when either of the two dc methods is selected, the **PostCheck** status instead acts as a **Check** status.

Comment

An optional user-specified data string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under

the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#).

Contingency Options Tab

The Options Tab enables you to control many parameters that govern how the contingency analysis flags violations, deals with violations that appeared in the Base Case, and documents the violations in the form of a report.

The bulk of the Options Tab contains five sub-tabs, each of which concerns a different aspect of the contingency analysis. The sub-tabs are:

- [Modeling](#)
- [Advanced Limit Monitoring](#)
- [Advanced Modeling](#)
- [Monitoring Exceptions](#)
- [Report Writing](#)
- [Miscellaneous](#)

Please see the corresponding help sections for assistance with each tab. Once you have finished setting the options for the contingency analysis, click **Set Options**. If, at any time, you wish to revert to the set of options that were defined the last time you clicked **Set Options** (or to the default options if you have never clicked **Set Options**), click **Reset**.

Contingency Options Tab: Modeling

Calculation Method

The calculation method defines how the power flow is solved during the contingency analysis. By default, Simulator uses a **Full Power Flow** for each contingency. For a large set of contingencies and a large case, this can take some time to complete. The Full Power Flow option can utilize either an AC solution or DC solution depending on whether or not the Power Flow Solution Options are set to use the [DC Option](#). Alternatively, the contingency analysis can be done by calculating the linear approximation of the impact of the contingencies by choosing either the **Linearized Lossless DC** or **Linearized Lossless DC with Phase Shifters** method. These two methods will both solve a set of contingencies much faster than the full power flow. The Lossless DC methods utilize sensitivities of devices to calculate the load flow in a linear fashion. The only difference between the two lossless DC methods is that the first method treats all phase shifters as free-flowing, while the method with phase shifters will hold all in-service phase shifters at their present MW flow value.

Limit Monitoring Settings

Click this button to open the [Limit Monitoring Settings Dialog](#) to change the limit settings for monitored elements. See [Limit Monitoring Settings](#) for more details.

For DC methods, allow amp limits by assuming a constant voltage magnitude

If a Lossless DC calculation method has been selected from the [Modeling Tab](#), this option will become available. If checked, Simulator will allow converting MVA limits to Amp limits by assuming constant voltage magnitudes based on the state operating point just prior to the contingency calculations. Converting to amp limits is only enforced if the [Limit Monitoring Settings Option of Treat Line Limits as Equivalent Amps](#) is checked. When this option is checked, the contingency loading of the line will be calculated from the post-contingency current on the line and the calculated amp limit.

Retry solution using the Robust Solution Process after a contingency solution failure

Checking this option will force the contingency analysis routine to attempt a robust solution following the failure of a standard Newton-Raphson solution. The robust solution attempt does not guarantee convergence, but will attempt to slowly approach a convergence solution if possible. This process is described in more detail under [Simulation Control](#).

Note: The Robust Solution Process is not used regardless of how this option is set when solving contingencies as part of either the PV or QV analysis available with the [PVQV add-on](#).

Use specific solution options for contingencies

When checked, Simulator will use a different set of [Solution Options](#) when solving the contingencies defined in the contingency set. To define the solution options used during the contingency analysis, click the **Define Solution Options** button to open the [Contingency Solution Options Dialog](#).

Do Not Use Post Power Flow Solution Action List

Checking this option prevents any globally defined post power flow solution actions defined with the case from being evaluated and performed in the post-contingency power flow solution during contingency analysis runs.

Make-Up Power

When solving a contingency, the make-up power defines how the post-contingency solution accounts for the change in system losses, generation, and load. There are three options for this make-up power.

Determine make-up using:

Area Participation Factors specified below

Simulator models this by temporarily switching to [Island-Based AGC](#) and using the **Calculate Participation Factors from Area Make Up Power Values** option during the contingency solution. The values specifying the **CTG Make Up Gen** for each area are used to determine the contribution of each area in an island to the amount of make-up power needed. Within each area, the individual participation factors of generators on AGC determine how much power will come from each generator.

Note that if all areas have **CTG Make Up Gen** equal to 0.0, then all make-up power will come from the island slack bus. This option generally yields faster solution times, but a concentration of make up power at the island slack bus may not accurately represent the response of the system following a contingency. The [Calculate Participation Factors from Area Make Up Power Values section of Island-Based AGC](#) describes this in further detail.

Generator Participation Factors From Entire Case Directly

Simulator models this by temporarily switching to [Island-Based AGC](#) and using the **Use Participation Factors of Individual Generators** option during the contingency solution. The individual participation factors of each generator within each island are used to determine how each generator meets the make-up power needs of the island. Area and super area interchange (ACE) requirements are ignored, and all generators on AGC are dispatched to account for the changes in system losses, generation, and load. This option is usually the best approximation of post-contingent system behavior, but may yield longer solution times than island slack bus make-up power (**Area Participation Factors specified below** with **CTG Make Up Gen** set to zero for all areas).

Same as Power Flow Case

This uses the area interchange options specified in the normal power flow.

Make-up Power Tolerance

This specifies the tolerance of the [Island-Based AGC](#) used for both the **Area Participation Factors specified below** and **Generator Participation Factors From Entire Case Directly** options.

Contingency Options Tab: Limit Monitoring

The Advanced Limit Monitoring Tab allows you to shape how limit violations are detected and reported.

Never report violations if...

This section controls the reporting of violations that should NEVER be reported. Minimum changes in branch flows, voltages, and interface flows which must be met before a device is reported as violating a limit can be specified. These options will only be used if the checkbox is checked at the beginning of this section. The minimum change values may be specified for:

Increase in line/transformer flows – This is the minimum change in percentage points that the loading on a line/transformer must increase before the line/transformer gets reported as a violation. For example, if this value is set to 2%, line limits are being monitored at 100%, and a line has a base case loading of 99% and a post-contingency loading of 100%, the line will not get reported as a violation.

Decrease in low bus voltage – This is the minimum change in a bus voltage that must occur for a bus low voltage violation to be reported. For example, if this value is set to 0.05 pu, the low voltage limit on a bus is 0.90 pu, the base case voltage at the bus is 0.91 pu, and the post-contingency voltage at the bus is 0.89 pu, this bus will not be reported as a low voltage violation.

Increase in high bus voltage – This is the minimum change in a bus voltage that must occur for bus high voltage violation to be reported. For example, if this value is set to 0.05 pu, the high voltage limit on a bus is 1.1 pu, the base case voltage at the bus is 1.09 pu, and the post-contingency voltage at the bus is 1.11 pu, this bus will not be reported as a high voltage violation.

Increase in interface flows – This is the minimum change in percentage points that the loading on an interface must increase before the interface gets reported as a violation. For example, if this value is set to 2%, interfaces are being monitored at 100%, and an interface has a base case loading of 99% and a post-contingency loading of 100%, the interface will not get reported as a violation.

Always report as a violation if...

This section allows you to specify the minimum change in flow or voltage at which point any device meeting the minimum change requirement will ALWAYS be reported, EVEN if the actual device limit (flow or voltage) is NOT violated. In other words, these options allow the reporting of large changes in flow or voltage, even if the device's actual limit is NOT itself violated. These options will only be used if the checkbox is checked at the beginning of this section. The minimum change values may be specified for:

Increase in line/transformer flows – This is the minimum change in line/transformer flow in percentage points that the loading on a line/transformer must increase so that the line/transformer gets reported as a violation even if the loading does not exceed the element's limit. For example, if this value is set to 2%, line limits are being monitored at 100%, and a line has a base case loading of 50% and a post-contingency loading of 63%, this line will be reported as a violation even though the post-contingency loading does not exceed the limit.

Decrease in low bus voltage – This is the minimum amount that a bus voltage must decrease for a bus low voltage violation to be reported even if the resulting voltage is higher than the bus low voltage limit. For example, if this value is set to 0.05 pu, the low voltage limit at a bus is 0.90 pu, the base case voltage at the bus is 1.0 pu, and the post-contingency voltage at the bus is 0.95 pu, this bus will be reported as a low voltage violation.

Increase in high bus voltage – This is the minimum amount that a bus voltage must increase for a bus high voltage violation to be reported even if the resulting voltage is lower than the bus high voltage limit. For example, if this value is set to 0.05 pu, the high voltage limit at a bus is 1.10 pu, the base case voltage at the bus is 1.0 pu, and the post-contingency voltage at the bus is 1.05 pu, this bus will be reported as a high voltage violation.

Increase in interface flows – This is the minimum change in interface flow in percentage points that the loading on an interface must increase so that the interface gets reported as a violation even if the loading does not exceed the interface limit. For example, if this value is set to 2%, interface limits are being monitored at 100%, and an interface has a base case loading of 75% and a post-contingency loading of 77%, this interface will be reported as a violation even though the post-contingency loading does not exceed the limit.

Caution should be used when using the **Always report...** options because this may result in a very large number of reported violations.

Report changes in bus dV/dQ sensitivity

Check this option to enable reporting changes in voltage to reactive power sensitivity for buses in the post-contingency solution. Set the sensitivity multiplier to the minimum change in sensitivity for reporting the value(s) under contingency. Any sensitivity that changes by the given multiple will be reported. You can also define an [advanced bus filter](#) using the Define Filter button, and choose either the newly defined bus filter or a previously existing bus filter for reporting dV/dQ sensitivity changes at only buses that meet the defined filter.

Also, regardless of the sensitivity multiple setting, any bus that meets the filter and has a negative dV/dQ will be reported as a violation.

Caution should be used when selecting this option. Calculating the dV/dQ sensitivities requires more computation time. Define a filter so that the dV/dQ sensitivities are only calculated for those buses for which it is important to calculate these values.

Report as a violations if a bus becomes disconnected

When checked, buses that become disconnected from the system due to a contingency will be reported as violations.

Re-reporting of base case violations

This section controls the reporting of Base Case violations. Because the concern of contingency analysis often is to identify those limitations that result directly from a particular outage or event, you may desire not to report all violations that were present in the Base Case with each contingency-specific set of violations. These options allow you to specify just how much of the Base Case violation information to report for each contingency. It gives you three options:

Do not report Base Case violations

When this option is checked, any element that was violated in the Base Case is omitted from the set of violations listed for each contingency.

List all Base Case violations for all contingencies

When this option is checked, all elements that were violated in the Base Case and are still violated post-contingency are included in the set of violations listed for each contingency.

Use these criteria

When this option is checked, only those elements that were violated in the Base Case and that meet the four criteria listed below will be listed with the contingency-specific violations. The four criteria include:

Minimum % increase in line/transformer flows: Only those branches that were violated in the Base Case whose flow has increased by at least this amount as a result of the contingency will be listed as contingency violations.

Minimum per-unit decrease in low bus voltage: Only those bus voltages that were violated in the Base Case that have decreased by at least this amount as a result of the contingency will be listed as contingency violations.

Minimum per-unit increase in high bus voltage: Only those bus voltages that were violated in the Base Case that have increased by at least this amount as a result of the contingency will be listed as contingency violations.

Minimum % increase in interface flows: Only those interfaces that were violated in the Base Case whose flow has increased by at least this amount as a result of the contingency will be listed as contingency violations.

When calculating the percentage for the Base Case violations used in the comparisons, the post-contingency rating sets are used. Comparisons made using percentage changes for branches are done using percentage points and not as a percentage of the actual change. For example, if the value for the minimum percent increase in line flow is 2%, lines are monitored at 100%, and a line is loaded at 100% in the base case and, the line must be loaded to at least 102% post-contingency to be reported.

How to Monitor Voltage Changes

Voltage changes can be monitored either directly on the amount of the bus per unit voltage, or based instead on a percentage change in per unit voltage from the base case per unit voltage values.

Contingency Options Tab: Advanced Modeling

Contingency Blocks and Global Actions

A [contingency block](#) stores a list of contingency actions and has a name (or label) associated with it. A contingency block can then be called from a contingency record. This allows you to define a block of common actions you wish to have processed during several different contingencies, and then assign the block to each contingency instead of constantly redefining the same actions for each contingency.

A [global action](#) is an action that will automatically be processed as part of EACH contingency. Thus if you have actions that are to be performed in every single contingency you define, you can insert it once in the global actions list, and Simulator will automatically use the defined action for every contingency it processes.

Model Expressions

Clicking this button brings up a list of the [model expressions](#). You may then insert a model expression by right-clicking on the list and choosing Insert. This will bring up the Define [Model Custom Expressions](#) dialog. Model Expressions may be used as part of Model Conditions.

Model Conditions

Clicking this button allows you to define [model conditions](#). Model Conditions may be used as part of the Model Criteria defined with [Contingency Elements](#).

Model Filters

Clicking this button allows you to define [model filters](#). Model Filters may be used as part of the Model Criteria defined with [Contingency Elements](#).

Generator Post-Contingency AGC

Clicking this button opens a dialog that allows the post-contingency AGC response of individual generators to be set. Set the **Post-CTG Prevent AGC Response** field to indicate whether or not a generator should be on AGC post-contingency. By default, generators are NOT set to prevent AGC response post-contingency. For the post-contingency participation factor field, **Post-CTG Part. Factor**, a value of *same* indicates that the generator should use the same participation factor as set with the generator record. Otherwise, set a numerical value for the post-contingency participation factor to use a factor other than the normal participation factor.

Bus Load Throw Over Records

Clicking this button allows you to define load throw over records in the [Bus Load Throw Over Records](#) list display.

Generator Maximum MW Responses in Post-Contingency

Clicking this button opens a list of generators, with a column, **CTG Max Response MW**, for entering the contingency maximum response.

During the post-contingency power flow solution, a user may enter a MW amount specifying the maximum amount of generator response from a generator. By default, these values are blank. If you enter a value in this column, then the generator response will be limited to this absolute MW response (response may be either an increase or a decrease).

Note: The user may also add a column called **CTG Max Response %** to the generator display. When entering data in this column, the **CTG Max Response MW** values will be set at a respective percent of the maximum MW output of the generator. If a generator's maximum MW output is less than or equal to zero, then the Maximum Response will always be set to zero.

Generator Line Drop and Reactive Current Compensation

Clicking this button opens a list of generators, with a column, **XLDC_RCC**, for entering line drop and reactive current compensation data.

Line Drop and Reactive Current Compensation represent an alternative method for performing generator voltage control. While on LDC/RCC control, the generator will vary its MVAR output in a manner that maintains the bus voltage at a fictitious bus that is a user-specified electrical impedance of **XLDC_RCC** away from the generator. This is called Line Drop Compensation when the impedance specified is positive, and Reactive Current Drop Compensation when the impedance specified is negative. If the absolute value of **XLDC_RCC** is less than or equal to $2 \times 10^{-6} \times \text{MVA Base}$, then the generator will regulate its terminal bus. (For a typical case with an MVA Base of 100 MVA, this value is 0.0002.)

The impedance is specified by entering a value for **XLDC_RCC** and the setpoint voltage is the same as used when regulating a generator in the more traditional manner. A generator will perform LDC/RCC control when it meets the following conditions:

- AVR = YES
- Use LDC_RCC = YES

Please note that if any generators at a bus are set to **Use LDC_RCC**, then this action will disable all traditional AVR control for generators at that bus. Other generators operating on LDC/RCC control are allowed, but no traditional AVR.

As a third special setting, you may also set the field **Use LDC_RCC** to *PostCTG* for use in contingency analysis. When a generator is set to *PostCTG*, while implementing the post-contingency power flow, the generator will change the **Use LDC_RCC** value to *YES*, thereby activating this new voltage control method for the generator. After [the reference state](#) is restored in the contingency analysis however, the generator will return back to a setting of *PostCTG*.

Post-Contingency Auxiliary File

The auxiliary file specified here is loaded at the start of each contingency (except for contingencies that have a [post-contingency auxiliary file specific to that contingency](#)). In this way, very specialized post-contingency settings can be specified. An example use of this feature could be changing the Generator voltage setpoints or AVR status for the post-contingency solution or changing generator AGC status for the post-contingency.

Note: Only data stored with the contingency reference state will be "reset" when the reference state is restored. Therefore, only data stored with the reference state should be loaded via a post-contingency auxiliary file. Click [here](#) for details on the specific information stored with the reference state.

Model reactive power for DC methods by...

When you choose to use one of the Lossless DC calculation methods, you can also specify how to handle changes in reactive power during the calculations. The lossless DC methods are based on the real power MW in the system, thus an assumption needs to be made about the reaction of the Mvar flows during the linear calculations.

The choices are:

Ignoring reactive power

Reactive power is completely ignored. This results in the MW flow only being compared to the limits of the elements in the system. It is important to recognize this fact, as branch and transformer limits are usually given in complex power (MVA) ratings, thus, ignoring reactive power results in comparing active power flow (MW) to total complex power limits (MVA.).

Assuming constant voltage magnitude

One way to include reactive power in the linearized DC results is to assume the voltage magnitudes remain constant during the linearized DC contingency analysis. Thus, the MW flows are determined from the linearized calculations, and the MVAR flows are calculated from the resulting flows and constant voltage magnitudes. Thus you will still receive an approximate complex power flow on each element (MVA), which can then be directly compared to the complex power limit of the element.

Assuming reactive power does not change

Another way to include reactive power in the linearized DC results is to assume that the reactive power magnitudes remain constant during the linearized analysis. Thus, the MW flows are determined from the linearized calculations, and the complex power flow of each element can be approximated using the calculated MW flows and the assumed constant Mvar flows from the base case. This again allows the approximate complex flow on each element to be compared to the complex power limit of the element.

Model Condition and Filter options

This section allows you to choose how assumptions about [Model Conditions](#) and [Model Filters](#) are made during the contingency analysis. Assumptions about the Model Conditions and Model Filters will impact how the [Model Criteria](#) are evaluated for contingency elements. You can specify to **Verify Model Conditions and Filters, Assume Model Conditions and Filters are TRUE**, or **Assume Model Conditions and Filters are FALSE**.

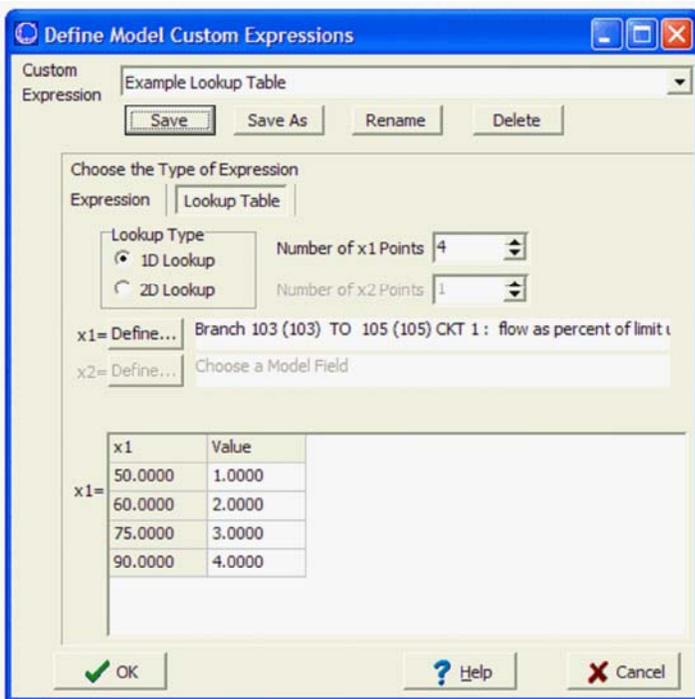
Model Expressions

Model Expressions can be accessed and created by choosing **Case Information and Auxiliary > Custom Model Expressions** in the [Model Explorer](#). To open the Define Model Custom Expressions dialog, right-click and choose **Insert**, when creating a new expression, or **Show Dialog** to view an existing expression.

The **Save**, **Save As**, **Rename**, and **Delete** buttons and the drop-down list of expressions at the top of the dialog allow for the additional of new expressions, modification of existing expressions, and removal of existing expressions. There are two types of Model Expressions: an expression and a lookup table.

Lookup Tables

The dialog as configured when creating a lookup table appears as follows:



A lookup table may be either one or two-dimensional by specifying the **Lookup Type**. The **Number of Points** must be set for each dimension. This will determine the size of the table. A Model Field must also be selected for each dimension. Clicking the **Define** button next to either the **x1** or **x2** field opens the Model Field Dialog. This dialog allows the selection of an **Element Type** and a specific element of this type. The list of elements behaves the same as the [Find Dialogs](#) found throughout Simulator. After selecting the element, choose the field associated with this element that will be used in the Lookup Table. Click **OK** to accept the element and field selected and close the dialog.

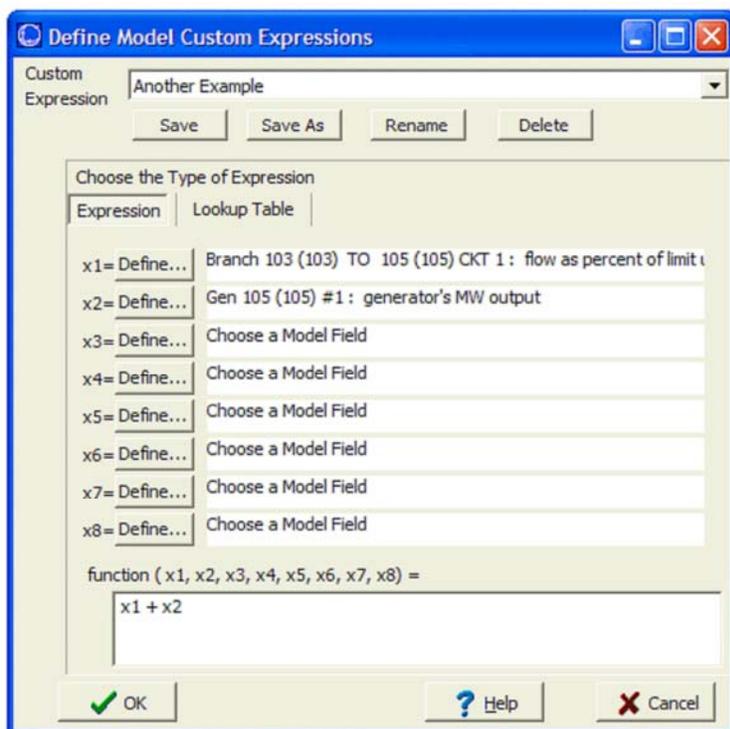
Fill in the Lookup Table with the desired values. If selecting a one-dimensional table, there will be two columns, one containing the values for the **x1** Model Field and the other containing the Values to return from the Lookup Table when the Model Field is at the defined value. There will be one header row plus as many additional rows as defined in the **Number of x1 Points**. If selecting a two-dimensional table, there will be one column for the **x1** Model Field and as many other columns as selected for the **Number of x2 Points**. There will be one header row for defining the **x2** values plus as many additional rows as defined in the **Number of x1 Points**. The cells under the first row

and to the right of the first column are used for defining the Values to return from the Lookup Table when the Model Fields meet the defined values.

If selecting a one-dimensional table, the returned Value is determined based on the current value of the **x1** Model Field. The returned value is the Value corresponding with the **x1** value that is less than or equal to the current value of the Model Field. If the current value of the Model Field is less than the smallest value of **x1** in the table, then the Value returned corresponds to the smallest **x1** value. If selecting a two-dimensional table, the same rules apply except that the Value returned corresponds to the current value of the **x1** AND **x2** Model Fields and where these intersect in the table.

Expressions

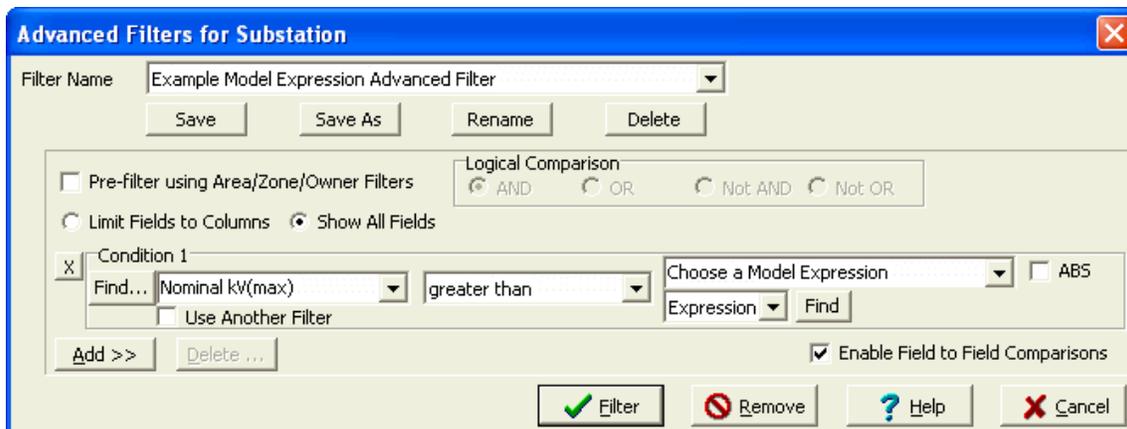
The dialog as configured when creating an expression appears below:



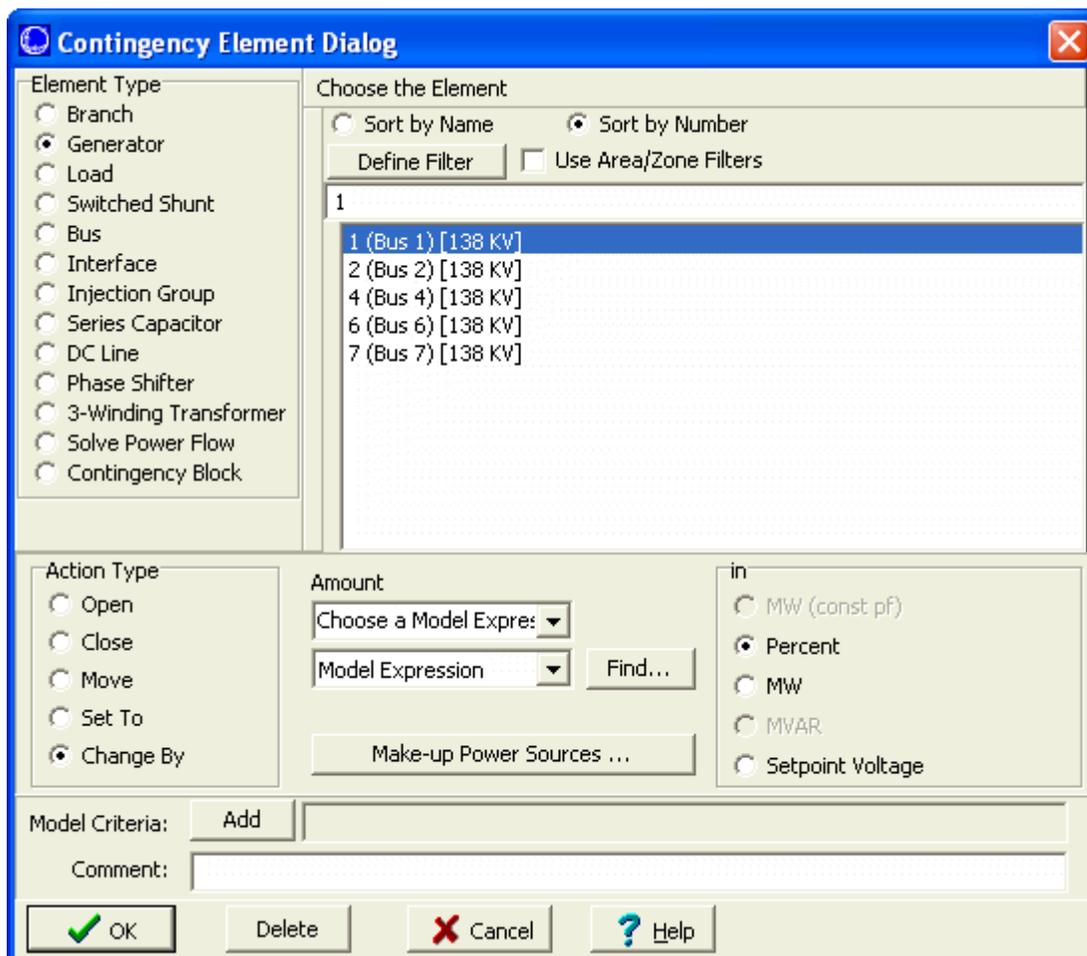
To define the Model Expression as an expression, specify the Model Fields that the expression should be a function of by clicking the **Define** button next to the appropriate variable. Clicking the **Define** button opens the Model Field Dialog. This dialog allows the selection of an **Element Type** and a specific element of this type. The list of elements of a particular type can be searched, filtered, and sorted to make finding an element easier. After selecting the element, choose the field associated with this element that will be used in the expression. Click **OK** to accept the element and field selected and close the dialog. After the Model Fields have been defined, enter the function description. See [Custom Expressions](#) for more information about defining expressions.

After model expressions have been defined, new options will appear on the [Contingency Element Dialog](#) and on the [Advanced Filter Dialog](#).

Normally, you may only enter a constant for the comparison value on the [Advanced Filter Dialog](#). However, if you select **Enable Field to Field Comparisons**, the advanced filter dialog will feature a drop-down from which you can choose *Expression*. After choosing *Expression*, you may select the name of the model expression, or click the **Find** button to search for the name.



Similarly, normally you may only enter a *Constant* or a *Field* for the *Move*, *Set To*, or *Change By Action Types* on the [Contingency Element Dialog](#). However, now you will have the option to choose *Model Expression*. To change the type of the **Amount**, choose *Model Expression* from the second drop-down box. After doing this, enter or choose the name of the model expression in the first **Amount** drop-down box, or click on the **Find...** button to search for a name.



Bus Load Throw Over Records

The Bus Load Throw Over list display, which is opened from the [Advanced Modeling](#) tab of the Contingency Dialog's Options page, provides you with the capability to define how load at a bus should be transferred to a different bus if the original terminal bus becomes disconnected from the system. This is referred to in Simulator as "throw over." This tool is most useful when performing contingency analysis scenarios in which buses containing loads become disconnected and you wish to analyze the impact on the system of switching the load from the disconnected bus to another bus that is still in service.

Load throw over will only attempt to move the load once, from the original bus to the load throw over bus. If the load throw over bus is already disconnected from the system, then the load will be treated as dropped during the contingency solution.

NOTE: The load throw over is only used when running the contingency analysis tool to analyze contingency effects on the system. Load throw over records are not used during manual solution of the power flow, even if you manually disconnect a bus with load and perform a load flow solution.

The Bus Load Throw Over display has the following fields:

Number, Name

The bus number and name of the load's terminal bus.

Nom kV

The nominal voltage level of the load's terminal bus.

Load Throw Over Bus Number

Enter the number of the bus you wish to have the load transferred to, should the original load terminal bus become disconnected during a contingency in the contingency analysis. The **Load Throwover Bus Name_kV** field will be automatically populated.

Load Throw Over Bus Name_kV

Enter the bus name and nominal kV (separated by a _ between the name and nominal kV) of the bus you wish the load to be transferred to, should the original load terminal bus become disconnected during a contingency in the contingency analysis. The **Load Throw Over Bus Number** will be automatically populated if the corresponding Bus Name_kV is found.

If choosing to identify objects by [labels](#), this field can also be used to display and enter the label of the load throw over bus.

Contingency Options Tab: Report Writing

Simulator can produce a report that details the results of the contingency analysis (see [Contingency Analysis Summary Tab](#) to see how to generate the report). The Report Writing Tab allows you to control the content and appearance of the report. By default, the report will identify each contingency, whether or not it could be solved, and what violations resulted from it. By selecting options on this tab, you can include additional information in the report.

Optional Report Contents**Case Summary**

The case summary prints the [Case Description](#) and then tallies the number of different power system components in the model.

Option Settings

If this item is checked, the report will list each of the options selected on the [Options Tab](#) of the Contingency Analysis Dialog.

Monitored Areas, Monitored Zones

If either of these items is checked, the report will identify the areas and/or zones in which Simulator has looked for limit violations, and over what voltage ranges.

Line Flow, Interface Flow, and Bus Voltage Extremes

If one of these items is checked, the report will list the worst-case line flows or voltages seen for each monitored element during the contingency analysis. None, one, two, or all three of these can be chosen at one time.

Base Case Outages

If this item is checked, the report will list the limit violations that existed in the Base Case.

All

Selecting this option will select all of the other options.

Identify buses by

This setting determines how the buses are listed in the data stored in the report. You can choose to have the buses displayed by number or name only, or by a combination of the number and name. You can also choose to identify with nominal voltage by checking the box labeled **Identify with Nominal Voltage**.

Show the actions involved in each contingency

If this box is checked, the definition of each contingency will be included in the report. The definition of each contingency simply identifies the actions that were implemented as part of the contingency.

Report only contingencies that cause violations

Checking this option will cause only the contingencies that cause violations to be shown in the report. Any contingency that did not cause a violation will not be included in the report. This option is useful if you wish to limit the size of the contingency report.

Report only limit type with violations for each contingency

Checking this option will result in reporting only the contingency violations of the type checked in the **Limit Type Violations to Include** box.

Report Inactive Violations and show all Rating Sets

When this box is checked, all violations that are normally being ignored during the contingency analysis (for example, base case violations) will be written to the report. Included with this option is the ability to show the different rating sets for each violated element.

Limit Type Violations to Include

Check the boxes of the types of violations you would like to be written to the report. This corresponds to the **Report only limit type with violations for each contingency** option above.

Maximum Violations of a single type to report

Enter the value of the maximum number of violations of a single type to be written to the report.

Create database-friendly tables

Checking this box will create three additional files for viewing contingency results: FILENAME_ctgelem, FILENAME_ctgviol, and FILENAME_ctgstat where FILENAME is the name of the file where the main report is saved. The user can choose which symbol to use to separate the columns by choosing a delimiting symbol. These files can be easily imported into a database or spreadsheet program such as Access or Excel as delimited text files, where they can be analyzed more rigorously. These files will be automatically created and saved in the same folder as the main report.

Produce Report

Clicking this button will open a Save Dialog for selecting the file to which the report should be written. The file is saved as a text file (*.txt). Enter the file path and name and click **Save** to write the report to the selected file. After saving the file, a dialog will provide the option of viewing the report file immediately.

Contingency Options Tab: Miscellaneous

Always save results with the contingency list when you save it to a file

If this option is checked, the violation results for each contingency will also be stored in an auxiliary file when the list of contingencies is saved to an auxiliary file. This allows for recovering the results of a contingency analysis run for a case, without reloading and re-running all the contingencies saved in the file.

Save contingency analysis definitions/results in the case PWB file

If this option is checked, the contingency definitions and any processed results will be saved with the load flow case in the PWB file when you save the case.

Setting Reference Case when Contingency Analysis is Opened

These options determine how the reference case for the contingency analysis is treated each time you re-open the contingency analysis. The first time you open the contingency analysis tool for a particular power flow case during a Simulator session, Simulator stores the current state of the power flow case as the reference case for the contingency analysis results. (See [Contingency Case References - Defining the Reference State](#) for more information.) However, each subsequent time you open the contingency analysis dialog in the same Simulator session, Simulator needs to know how to set the reference state for the case. The choices are:

Always set reference case to the current case

This option will always assume that any changes you have made to the load flow case since the contingency analysis was last opened should be applied and will store the current state of the load flow as the new reference state for the contingency analysis.

Always use the existing contingency analysis reference case

This option will assume that the original reference case stored when the contingency analysis tool was originally launched should always be the reference state for the contingency analysis. This means that any changes that have been made to the load flow case since the initial launch of the contingency analysis will be lost, as the contingency analysis tool will reset the load flow state to the state stored with the first contingency analysis instance.

Prompt for which reference case to use (the current case or the pre-existing reference case) whenever the Contingency Analysis Form is opened

When this option is checked, you will always be prompted when you re-open the contingency analysis following the initial instance. You will then have the option to choose from one of the two previous settings, to either set the reference state to the current case or use the existing reference case currently stored with the contingency analysis tool.

Contingency Summary Tab

The Summary Tab of the [Contingency Analysis Dialog](#) provides additional information on the status of the contingency analysis run and allows you to start, pause, resume, and abort the contingency run. The top half of the Summary Tab charts the progress of the contingency analysis run and issues warning messages when a particular contingency fails to solve. The next section features counters that indicate the total number of contingencies that comprise the list, the number of these contingencies that have been processed thus far, the number of contingencies that failed to solve, and the total number of violations that have been flagged. To start the contingency analysis, click **Start**. To pause the contingency analysis, click **Pause**. To resume a paused contingency run, click **Continue**. Finally, to abort the analysis, click **Abort**.

The **Refresh Displays After Each Contingency** checkbox is used to force a refresh of the counters after each contingency, which may slow down processing of the contingency set (accordingly, this box is unchecked by default to maximize solution speed).

Comparing Contingency Analysis Results

After performing an automated contingency analysis, the results of the analysis can be saved to an auxiliary file. Once the results have been saved to a file, they can then be compared to different results in another file or to results existing in memory in the Contingency Analysis tool. To compare two sets of contingencies, do the following:

- Process each set of contingencies and save the results for at least one set in an auxiliary file.
 - To save a contingency list, right-click on the list of contingencies and choose **Save As > Auxiliary File**.
 - In the Save Contingency File Dialog, choose a name for the file, and then click **Save**.
 - You will then be prompted to choose options for saving the contingency list to an auxiliary file. By default, the contingency definitions themselves are saved, along with the contingency options. The optional information you may choose to save are the [Limit Monitoring Settings](#), General [Power Flow Solution Options](#), [List Display Settings](#), and [Contingency Results](#). When saving the contingency results with the file, you may also choose to include inactive violations.
Inactive violations are considered violations on elements for a DIFFERENT limit than what is currently being monitored. For example, a branch may have an A limit rating of 50 MVA, and a B limit rating of 100 MVA. Consider if the B limits are being used to report violations during a contingency. A value of 75 MVA flow on the branch would not be reported in the contingency analysis as a violation, considering the B limit of 100 MVA is being used. However, Simulator internally will flag the element as a potential violation if the limit set used is switched to the A rating set. Simulator considers these types of situations as inactive violations. These are kept track of to allow the user to easily switch the rating set used for reporting violations from one set to another and see the results immediately, without having to re-run the entire contingency set to determine the violations for the new rating set. Choosing to include the inactive violations when saving an auxiliary file maintains this flexibility when the contingency definitions and results are read into a case from an auxiliary file. You can also choose for the identifiers used in the file to be either the bus numbers or the bus name and nominal kV voltage.
Note: For comparing two lists of contingency results, you MUST save the contingency results with each of the two auxiliary files being compared.
 - Click **OK** to save the contingencies and the results to the auxiliary file specified.
- Once you have two different lists of contingency results (at least one of the lists must be saved in an auxiliary file), right-click on the contingency list and choose **Compare Two Lists of Contingency Results** or choose **Other > Compare Two Lists of Contingency Results**. This will bring up a dialog on which you have to specify the Contingency Lists you are interested in comparing. You must specify the **Controlling Contingency List** and the **Comparison Contingency List**. The definitions of these two lists are found below.
- Click on the **Browse** buttons to specify the two Contingency Lists you would like to compare. You can also choose to use the presently open Contingency List as either the Controlling Contingency List or the Comparison Contingency List.

Controlling Contingency List:

The list that controls what is displayed on the dialog. Only contingencies that are defined in this list will be displayed on the form. Only violations that occur for contingencies in this list will appear in the Violations List for each contingency.

Comparison Contingency List:

This is the list to which the Controlling Contingencies will be compared. Comparisons will occur for those contingencies in the lists that have the same **CONTINGENCY NAME**. Note that contingencies in the Comparison list whose **CONTINGENCY NAME** does not match one of those in the Controlling list will not be displayed. Also, violations which occur in a specific Comparison contingency that do not occur in the respective Controlling contingency will not be displayed.

Example:

A user has a power system case and a list of contingencies. The user runs contingency analysis on this system for this list of contingencies. The results are saved in a file called comparison.aux. The user now changes the system state, possibly adding in a 500 MW transaction between two areas. The contingency analysis is run on this new state of the system for the list of contingencies. The results are saved in a file called controlling.aux. You should define the contingency results you are more interested in viewing as the Controlling List because this list determines what is shown on the dialog. In this case, we are more interested in seeing the violations caused when the transaction is in place, so that list is defined as the Controlling List.

The comparison of the two sets of contingencies is now done by right-clicking and choosing **Compare Two Contingency List Results**. The file controlling.aux is set as the **Controlling List** and comparison.aux as the **Comparison List**.

- After clicking **OK** on the dialog, the contingency lists will be read from the specified files or from the presently open list. After Simulator has completed reading these files, a prompt will appear which asks, "**Would you like to set the dialog with default columns for comparing contingency lists?**". It is recommended that you choose **YES** so that the case information displays on the Contingency Dialog will automatically be set to show fields that will help you compare the two lists of contingencies. For information on the default fields used when comparing contingencies see [Comparing Contingencies List Displays](#).

Comparing Contingencies List Displays

When comparing two sets of contingency results, there are several additional default fields added to several list displays which help you compare the results. These fields are described below for the various list displays.

Contingencies Records on [Contingencies Tab](#)

Violations

The total number of violations for this contingency under the Controlling List.

Comp Violations

The total number of violations for this contingency under the Comparison List.

New Violations

The number of violations which occur in the Controlling List which do not occur in the Comparison List.

Max Branch % (Max Interface %)

The highest branch (interface) violation caused by this contingency in the Controlling List.

Comp Max Branch % (Comp Max Interface %)

The highest branch (interface) violation caused by this contingency in the Comparison List.

Worst Branch Violation (Worst Interface Violation)

This is the maximum of the following two values: [**Worst Increase Violation**] and [**Worst New Violation - 100%**]

Min Volt (Max Volt)

The worst violation in the controlling list.

Comp Min Volt (Comp Max Volt)

The worst violation in the comparison list.

Worst LowV Violation (Worst HighV Violation)

The worst new violation.

[Contingency Violations Display](#) on [Contingencies Tab](#) and Contingency Violations on the [Lines, Buses, Interfaces Tab](#)

Value

The value of the violation under the controlling list.

Comp Value

The value of the violation under the comparison list.

Diff Value

The difference between **Value** and **Comp Value**

Limit

The limit of the element in the controlling list.

Comp Limit

The limit of the element in the comparison list.

Diff Limit

The difference between **Limit** and **Comp Limit**

Percent

The percent violation in the controlling list.

Comp Percent

The percent violation in the comparison list.

Diff Percent

The difference between **Percent** and **Comp Percent**

Line/Transformer (Interface) Records on the [Lines, Buses, Interfaces Tab](#)

Violations

This shows the number of branch (interface) violations which occurred in the Controlling List

New Violations

This shows the number of branch (interface) violations which occurred in the Controlling List, but did not occur in the Comparison List.

Max % Loading Cont.

The worst branch (interface) violation in the controlling list.

Max % Ld Cont Comp

The worst branch (interface) violation which occurred in the comparison list.

Worst Increased Violation

The worst increase in a branch (interface) violation from the comparison list to the controlling list.

Bus Records on the [Lines, Buses, Interfaces Tab](#)

Violations

This shows the number of violations which occurred in the Controlling List

New Violations

This shows the number of violations which occurred in the Controlling List, but did not occur in the Comparison List.

Max Voltage Cont.

The worst high voltage violation in the controlling list.

Max Voltage Cont Comp

The worst high voltage violation which occurred in the comparison list.

Worst Max Volt CTG Change

The worst increase in a high voltage violation.

Min Voltage Cont.

The worst low voltage violation in the controlling list.

Min Voltage Cont Comp

The worst low voltage violation which occurred in the comparison list.

Worst Min Volt CTG Change

The worst decrease in a low voltage violation.

Nomogram Interfaces Records on the [Lines, Buses, Interfaces Tab](#)

Violations

This shows the number of violations which occurred in the controlling list.

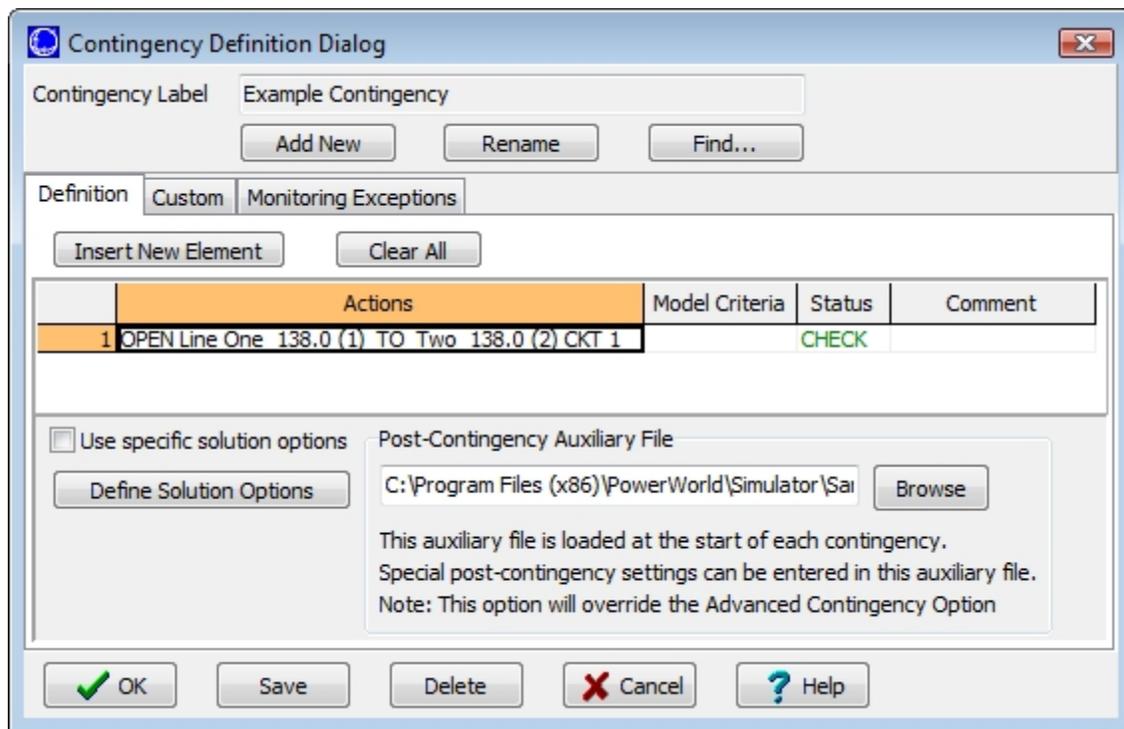
Max % Loading Cont.

The worst nomogram interface violation in the controlling list.

Contingency Definition Dialog

The Contingency Definition Dialog (shown below) serves as an information source for displaying the [Contingency Element](#) (or Elements) associated with individual [Contingencies](#) defined in the case. You can use the Contingency Definition Dialog to scroll through the list of elements, to view and modify their definitions, to insert new elements in a contingency or to delete a contingency. You may access this dialog by choosing either **Show Dialog** or **Insert** from the local menu of the [Contingency Records Display](#).

After making changes, click **OK** to save your changes and close the dialog. Click **Cancel** to close the dialog without saving your changes. Click **Save** to save your changes (including the addition of a new contingency) without closing the dialog (this allows you to keep working with the dialog). Click **Delete** to remove the selected contingency from the contingency list.



Contingency Definition Dialog

The Contingency Definition Dialog has the following controls that are available regardless of the tab that is currently selected:

Contingency Label

Identifies the name of the currently displayed contingency. Use the drop-down arrow to select a different contingency, or use the scroll buttons to navigate through the list of contingencies. When adding a new contingency, Contingency Label will show *New Contingency*. The user can change the Contingency Label by clicking **Rename**.

Add New

Click the **Add New** button to add a new [contingency](#) to the contingency list for the case. You will be prompted to enter a unique name for the new contingency. After naming the new contingency, the name appears in the Contingency Label and you can insert new [elements](#) in the contingency definition.

Rename

Allows you to rename the selected contingency.

Find

Opens a dialog that will allow the use of [advanced search methods](#) for finding a particular contingency.

Definition Tab**Insert New Element**

Click this button to add a new element to the contingency. This will open the [Contingency Element Dialog](#), used to define the Action, Model Criteria and Comment associated with the element. When you return to the Contingency Definition Dialog, the display will contain the newly inserted element.

Clear All

Removes all elements from the contingency definition. The Contingency Elements Table will then appear blank, indicating that the contingency involves no associated actions.

Definitions Display

The Contingency Definitions Display lists the [Elements](#) assigned to the selected [Contingency](#). Select **Insert** from the local menu or click on the **Insert New Element** button to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency. For more information about this display, see the [Contingency Definitions Display](#).

Define Solution Options

Click this button to open the [Contingency Solution Options Dialog](#), used to define specific power flow solutions options for use under the selected contingency.

Use Specific Solution Options

Check this box to enable the use of Contingency Specific Solution Options (see Define Solution Options above).

Post-Contingency Auxiliary File

The auxiliary file specified here will be loaded at the start of this contingency, which can be used to alter the reference state for this contingency. If an auxiliary file is specified here, then the contingency options [Advanced Modeling](#) Post-Contingency Auxiliary File will not be loaded for this contingency. Click the **Browse** button to open a file dialog for file selection. To remove the file, delete the contents of the text box.

Custom Tab

The [Custom page](#) of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the contingency. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the contingency. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Monitoring Exceptions Tab

Monitoring exceptions can be defined for each contingency. Defining exceptions is useful when specific elements need to be excluded or included for monitoring on a contingency-by-contingency basis. Click [here](#) for more information about monitoring exceptions.

Contingency Element Dialog

The Contingency Element Dialog provides information on the individual elements that comprise a contingency definition. You may use this dialog to modify an existing contingency's definition or to add elements to new or existing contingencies.

There are several ways to open the Contingency Element Dialog:

- By pressing the **Insert New Element** button on the [Contingency Definition Dialog](#).
- By right-clicking on a [Contingency Definition Display](#) and choosing **Insert** or **Show Dialog**.
- By right-clicking on the [Contingency Tab](#) of the [Contingency Analysis Dialog](#) and choosing **Insert Special > Quick Insert of Single Element Contingency**.

After making the desired changes, click **OK** to save changes and close the dialog or click **Cancel** to close the dialog without saving your changes.

Click **Delete** to remove the element from the contingency.

Contingency Element Dialog

Element Type

- Branch
- Generator
- Load
- Switched Shunt
- Bus
- Interface
- Injection Group
- Series Capacitor
- DC Line
- Phase Shifter
- 3-Winding Transformer
- Solve Power Flow
- Contingency Block

Choose the Element

Sort by Name Sort by Number

Define Filter Use Area/Zone Filters

Search For Near Bus	Select Far Bus, CKT
1 (One) [138 KV]	2 (Two) [138 KV] CKT 1
2 (Two) [138 KV]	3 (Three) [138 KV] CKT 1
3 (Three) [138 KV]	
4 (Four) [138 KV]	
5 (Five) [138 KV]	
6 (Six) [138 KV]	
7 (Seven) [138 KV]	

Action Type

- Open
- Close
- Move
- Set To
- Change By

Amount:

in

- MW (const pf)
- Percent
- MW
- MVAR
- Setpoint Voltage

Model Criteria:

Comment:

Contingency Element Dialog

The Contingency Element Dialog has the following controls:

Element Type

Indicates whether the element involved in the contingency action is a branch, generator, load, switched shunt, bus, interface, injection group, series capacitor, DC line, phase shifter, [contingency block](#) or solve power flow. The Element Type will dictate what **Action Types** are available, and the selection of the Element Type, in conjunction with the **Action Type**, **Amount**, and **in** options settings, determines what actually happens during the contingency.

What actually happens during a contingency action based on the **Element Type**, **Action Type**, and **in** options is described in detail below:

Branch

Open

The Open action will set the Status of the selected branch to *Open* if the branch Status is *Closed*. If the branch Status is already *Open*, then this action does nothing.

Close

The Close action will set the Status of the selected branch to *Closed* if the branch Status is *Open*. If the branch Status is already *Closed*, then this action does nothing.

Generator

Open

The Open action will set the Status of the selected generator to *Open* if the generator Status is *Closed*. If the generator Status is already *Open*, then this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Close

The Close action will set the Status of the selected generator to *Closed* if the generator Status is *Open*. If the generator Status is already *Closed*, then this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Move

The Move action allows the transferring of generation from an existing generator bus to another bus. A generator move is done on a bus basis and impacts the total generation of all generators at the selected bus. Generators that are adjusted by the Move action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation move will occur. Generation may be moved in **Percent** or **MW**. A **Percent** move will move the specified percent of the current generator MW and Mvar to the defined **Bus to Move to**. A **MW** move will move the specified amount of generator MW to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the output of the selected generator bus, and a negative amount will increase the output of the selected generator bus. The Bus to Move to will respond accordingly.

If a generator already exists at the bus to which the generation is being moved, the generation at that bus is adjusted according to the move amount. If a generator does not exist but a load exists, then the load is adjusted by the move amount. If no generator or load exists, then a load is added at the bus and the injection is set according to the move amount. In terms of power injection, positive generation is the same as negative load.

Set To

The Set To action will set the selected bus' generator parameters to a specified **Amount**. Generator Set To actions are done on a bus basis and impact the total generation of all generators at the selected bus. Generators that are adjusted by a Set To action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation change will occur. Generator parameters can be set in **Percent**, **MW**, or **Setpoint Voltage**. When the generation is set in **Percent**, total generator MW output will be set to a value based on the specified percent of the current total generator MW output. When the generation is set in **MW**, total generator MW output will be set to the specified MW amount. When the generation is set in **Setpoint Voltage**, the voltage set-point of all generators at the selected bus will be set to the specified amount in per unit.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Change By

The Change By action will change the selected bus' generator parameters by a specified **Amount**. Generator Change By actions are done on a bus basis and impact the total generation of all generators at the selected bus. Generators that are adjusted by a Change By action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation change will occur. Generator parameters can be changed in **Percent**, **MW**, or **Setpoint Voltage**. When the change is in **Percent**, total generator MW output will be changed based on the specified percent of the total current generator MW output. When the change is in **MW**, total generator MW output will be changed by the specified MW amount. When the change is in **Setpoint Voltage**, the voltage set-point of all generators at the selected bus will be changed by the specified amount in per unit.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Load

Open

The Open action will set the Status of the selected load to *Open* if the load Status is *Closed*. If the load Status is already *Open*, then this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Close

The Close action will set the Status of the selected load to *Closed* if the load Status is *Open*. If the load Status is already *Closed*, then this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Move

The Move action allows the transferring of load from an existing load bus to another bus. The load move is done on a bus basis and impacts the total of all loads at the selected bus. Load may be moved in **MW (const pf)**, **Percent**, **MW**, or **Mvar**. The load amount that is moved is the total actual load and may be different than the nominal load if the area of the load has a Load MW Multiplier other than 1.0 or the load contains constant impedance or constant current components. If there is no on-line load at the selected load bus or the bus at which the load is located is *Open*, no load move will occur. A **MW (const pf)** move will move the specified amount of load MW to the **Bus to Move to** while moving a Mvar amount that will maintain a constant power factor at the selected load bus. A **Percent** move will move the

specified percent of the present load MW and Mvar to the defined Bus to Move to. A **MW** move will move the specified amount of the load MW to the Bus to Move to. A **Mvar** move will move the specified amount of the load Mvar to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the load at the selected bus, and a negative amount will increase the load at the selected bus. The Bus to Move to will respond accordingly.

If a load already exists at the bus to which the load is being moved, the load at that bus is adjusted according to the move amount. If no load exists, then a load is added at the bus and the load is set according to the move amount.

Set To

The Set To action will set the selected bus' load parameters to a specified **Amount**. The load change is done on a bus basis and impacts the total of all loads at the selected bus. Load parameters can be set in **MW (const pf)**, **Percent**, **MW**, or **Mvar**. The load value that is set for each of these options is the actual constant power component of the load. This value may be different than the nominal constant power component of the load if the area that contains the load has a Load MW Multiplier other than 1.0. If there is no on-line load at the selected load bus and the bus at which the load is located is *Closed*, load status at the selected load bus will change so that one load is *Closed* and the total constant power load at the bus will be set to the specified amount. If the bus at which the load is located is *Open* or there is no existing load, either *Open* or *Closed*, at that bus, then no load change will occur. When the load is set in **MW (const pf)**, the actual constant power MW will be set to the specified amount and the actual constant power Mvar will be set so that a constant power factor is maintained at the load bus. When the load is set in **Percent**, the actual constant power MW and Mvar will be set to the specified percent of the present load MW and Mvar. When the load is set in **MW**, the actual constant power MW at the selected load will be set to the specified amount. When the load is set in **Mvar**, the actual constant power Mvar at the selected load will be set to the specified amount.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Change By

The Change By action will change the selected bus' load parameters by a specified **Amount**. The load change is done on a bus basis and impacts the total of all loads at the selected bus. Load parameters can be changed in **MW (const pf)**, **Percent**, **MW**, or **Mvar**. The load value that is changed for each of these options is the actual constant power component of the load. This value may be different than the nominal constant power component of the load if the area that contains the load has a Load MW Multiplier other than 1.0. If there is no on-line load at the selected load bus or the bus at which the load is located is *Open*, no load move will occur. When the load is changed in **MW (const pf)**, the actual constant power MW will be changed by the specified amount and the actual constant power Mvar will be changed so that a constant power factor is maintained at the load bus. When the load is changed in **Percent**, the actual constant power MW and Mvar will be changed by the specified percent of the present load MW and Mvar. When the load is changed in **MW**, the actual constant power MW at the selected load bus will be changed by the specified amount. When the load is changed in **Mvar**, the actual constant power Mvar at the selected load bus will be changed by the specified amount.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Switched Shunt

Open

The Open action will set the Status of the selected switched shunt to *Open* if the switched shunt Status is *Closed*. If the switched shunt Status is already *Open*, then this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in the real power portion of the switched shunt due to the contingency action.

Close

The Close action will set the Status of the selected switched shunt to *Closed* if the switched shunt Status is *Open*. If the switched shunt Status is already *Closed*, then this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in the real power portion of the switched shunt due to the contingency action.

Move

The Move action allows the transferring of switched shunt MW and/or Mvar from an existing switched shunt bus to another bus. Switched shunt moves are done on a bus basis and impact the total value of all switched shunts at the selected bus. During the move action, the nominal values of the switched shunts are adjusted, and switched shunts that are adjusted have their Control Mode set to *Fixed*. If there are no switched shunts at the selected switched shunt bus with a Status of *Closed*, no switched shunt move will occur. Switched shunt Moves may be done in **Percent**, **MW**, or **Mvar**. A switched shunt Move in **Percent** will move the specified percent of present nominal MW and Mvar at the switched shunt bus to the defined **Bus to Move to**. A **MW** move will move the specified amount of switched shunt MW to the Bus to Move to. A **Mvar** move will move the specified amount of switched shunt Mvar to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the nominal MW and/or Mvar at the selected switched shunt bus, and a negative amount will increase the nominal MW and/or Mvar at the selected switched shunt bus. The Bus to Move to will respond accordingly.

If a switched shunt exists at the bus to which the switched shunt MW and/or Mvar is being moved, the switched shunt at that bus is adjusted according to the move amount. If there are existing switched shunts at the Bus to Move to but all of them have a Status of *Open*, the first switched shunt Status will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal MW and/or Mvar will be adjusted according to the move amount. If no switched shunt exists at the Bus to Move to, then the nominal constant impedance load is adjusted according to the move amount. If no load exists at that bus, then a load is added and the nominal constant impedance load is adjusted according to the move amount.

Set To

The Set To action will set the selected bus' switched shunt parameters to a specified **Amount**. Switched shunt changes are done on a bus basis and impact the total nominal value of all switched shunts at the selected bus. Switched shunts that are adjusted during Set To actions have their Control Mode set to *Fixed*. Switched shunt parameters can be set in **Percent**, **MW**, **Mvar**, or **Setpoint Voltage**. When a switched shunt bus is set in **Percent**, the nominal MW and Mvar are set to the specified percent of the present nominal MW and Mvar of the switched shunt bus. When the switched shunt bus is set in **MW**, the nominal MW is set to the specified amount. When the switched shunt bus is set in **Mvar**, the nominal Mvar is set to the specified amount. When the switched shunt bus is set in **Setpoint Voltage**, the Target Value of the controllable switched shunt at the bus is set to the specified amount.

Switched shunts may either regulate voltage or generator Mvar. The amount entered for the Setpoint Voltage is considered to be in per unit when the switched shunt is regulating voltage and is considered to be in Mvar when regulating generator Mvar. If there are no switched shunts at the selected bus with a Status of *Closed* when setting MW or Mvar amounts, the Status of the first switched shunt at the bus will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal value will be set appropriately.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in switched shunt MW due to the contingency action.

Change By

The Change By action will change the selected bus' switched shunt parameters by a specified **Amount**. Switched shunt changes are done on a bus basis and impact the total nominal value of all switched shunts at the selected bus. Switched shunts that are adjusted during the Change By action have their Control Mode set to *Fixed*. Switched shunt parameters can be changed in **Percent**, **MW**, **Mvar**, or **Setpoint Voltage**. When a switched shunt bus is changed in **Percent**, the nominal MW and Mvar are changed by the specified percent of the present nominal MW and Mvar of the switched shunt bus. When the switched shunt bus is changed in **MW**, the nominal MW is changed by the specified amount. When the switched shunt bus is changed in **Mvar**, the nominal Mvar is changed by the specified amount. When the switched shunt bus is changed in **Setpoint Voltage**, the Target Value of the controllable switched shunt at the bus is change by the specified amount. Switched shunts may either regulate voltage or generator Mvar. The amount entered for the Setpoint Voltage change is considered to be in per unit when the switched shunt is regulating voltage and is considered to be in Mvar when regulating generator Mvar. If there are no switched shunts at the selected bus with a Status of *Closed* when setting MW or Mvar amount, the Status of the first switched shunt at the bus will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal value will be set appropriately.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in switched shunt MW due to the contingency action.

Bus

Open

The Open action will set the Status of all ac branches connected to the selected bus to *Open*. If all of the ac branches are already open, then this action does nothing.

Interface

Open

The Open action will set the Status of all ac branches in the selected interface to *Open*. If all of the ac branches are already open, then this action does nothing. The Status of any dc lines, generators, loads, or injection groups that are part of the interface will remain unchanged due to the contingency action. If the selected interface contains additional interfaces, the Status of all ac branches in the additional interfaces will also be set to *Open*.

Close

The Close action will set the Status of all ac branches in the selected interface to *Closed*. If all of the ac branches are already closed, then this action does nothing. The Status of any dc lines, generators, loads, or injection groups that are part of the interface will remain unchanged due to the contingency action. If the selected interface contains additional interfaces, the Status of all ac branches in the additional interfaces will also be set to *Closed*.

Injection Group

Open

The Open action will set the Status of all generators and loads in the selected injection group to *Open* if the Status is *Closed*. If the Status is already *Open*, this action does nothing. If the injection group contains other injection groups, the Status of the generators and loads in the other injection groups will also be set to *Open*.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Close

The Close action will set the Status of all generators and loads in the selected injection group to *Closed* if the Status is *Open*. If the Status is already *Closed*, this action does nothing. If the injection group contains other injection groups, the Status of the generators and loads in the other injection groups will also be set to *Closed*.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Set To

The Set To action will set the real power injection of the injection group to the specified **Amount**. If loads are included in the injection group, the Mvar load will be adjusted by keeping the power factor constant. The injection can be set in **Percent**, **MW**, **Percent (In Merit Order)**, or **MW (In Merit Order)**. When setting the injection in **Percent** or **Percent (In Merit Order)**, the real power injection is set to the specified percent of the present real power injection. The present real power injection of the injection group is determined by taking the difference between the total generator MW and total load MW in the injection group. When setting the injection in **MW** or **MW (In Merit Order)**, the real power injection is set to the specified amount. When selecting either the **Percent** or **MW** option, all generators and loads in the injection group are adjusted according to their relative participation factors. All generators and loads with non-zero participation factors will be adjusted to meet the desired injection. When choosing one of the **In Merit Order** options, only generators will be adjusted to meet the desired injection. Generators will be adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum limit before moving on to the next generator. This process continues until the desired injection is met.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Change By

The Change By action will change the real power injection of the injection group to the specified **Amount**. If loads are included in the injection group, the Mvar load will be adjusted by keeping the power factor constant. The injection can be set in **Percent**, **MW**, **Percent (In Merit Order)**, or **MW (In Merit Order)**. When changing the injection in **Percent** or **Percent (In Merit Order)**, the real power injection is changed by the specified percent of the present real power injection. The present real power injection of the injection group is determined by taking the difference between the total generator MW and total load MW in the injection group. When changing the injection in **MW** or **MW (In Merit Order)**, the real power injection is changed by the specified amount. When selecting either the **Percent** or **MW** option, all generators and loads in the injection group are adjusted according to their relative participation factors. All generators and loads with non-zero participation factors will

be adjusted to meet the desired injection change. When choosing one of the **In Merit Order** options, only generators will be adjusted to meet the desired injection. Generators will be adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum limit before moving on to the next generator. This process continues until the desired injection change is met.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Series Capacitor

Bypass

The Bypass action will change the Bypass Status of the selected series capacitor to *Bypassed* if the Bypass Status is *Not Bypassed*. If the Bypass Status is already *Bypassed*, this action does nothing.

Inservice

The Inservice action will change the Bypass Status of the selected series capacitor to *Not Bypassed* if the Bypass Status is *Bypassed*. If the Bypass Status is already *Not Bypassed*, this action does nothing.

Set To

The Set To action will set the series reactance of the series capacitor to a specified **Amount**. The reactance may be set in **X (percent)** or **X (per unit)**. If the reactance of the series capacitor is set in **X (percent)**, the per unit reactance is set to the specified percent of the present per unit reactance of the series capacitor. If the reactance is set in **X (per unit)**, the per unit reactance of the series capacitor is set to the specified amount.

DC Line

Open

The Open action will set the Control Mode of the selected dc line to *Blocked*. If the dc line is already blocked, then this action does nothing.

Close

The Close action will set the Control Mode of the selected dc line to either *Power* or *Current*. When selecting the Close action, the **Amount** must be specified for the new Setpoint of the dc line in either **MW** or **Amps**. If specifying the Setpoint in **MW**, the Control Mode of the dc line will be set to *Power*. If specifying the Setpoint in **Amps**, the Control Mode of the dc line will be set to *Current*.

Set To

The Set To action will set the selected dc line's Setpoint to a specified **Amount** and could possibly change the Control Mode of the line. The Setpoint can be set in **Percent**, **MW**, or **Amps**. When the dc line's Setpoint is set in **Percent**, the Setpoint is set to the specified percentage of the present Setpoint of the line and the Control Mode remains unchanged. If the Control Mode is presently *Blocked*, the resulting Setpoint will be 0 regardless of the Amount entered. When the Setpoint is set in **MW**, the Setpoint is set to the specified MW amount, and the Control Mode is changed to *Power* if the Control Mode is presently *Current*. If the Control Mode is *Blocked* or *Power*, then the Control Mode remains unchanged. When the Setpoint is set in **Amps**, the Setpoint is set to the specified current amount, and the Control Mode is set to *Current* if the Control Mode is presently *Power*. If the Control Mode is *Blocked* or *Current*, then the Control Mode remains unchanged.

Change By

The Change By action will change the selected dc line's Setpoint by the specified **Amount** and could possibly change the Control Mode of the line. The Setpoint can be changed in

Percent, MW, or Amps. When the dc line's Setpoint is changed in **Percent**, the Setpoint is changed by the specified percentage of the present Setpoint of the line and the Control Mode remains unchanged. If the Control Mode is presently *Blocked*, the resulting Setpoint will be 0 regardless of the Amount entered. When the Setpoint is changed in **MW**, the Setpoint is changed by the specified MW amount, and the Control Mode is changed to *Power* if the Control Mode is presently *Current*. If the Control Mode is *Blocked* or *Power*, then the Control Mode remains unchanged. When the Setpoint is changed in **Amps**, the Setpoint is changed by the specified current amount, and the Control Mode is set to *Current* if the Control Mode is presently *Power*. If the Control Mode is *Blocked* or *Current*, then the Control Mode remains unchanged.

When the Control Mode changes from either *Power* to *Current* or from *Current* to *Power*, the appropriate conversions are done so that the resulting Setpoint value reflects the original Setpoint and the change in Setpoint in the same units.

Phase Shifter

Set To

The Set To action will set the middle of the regulation range of the selected phase shifter to the specified **Amount**. The middle of the regulation range is calculated by taking the average of the Regulation Minimum MW Flow and Regulation Maximum MW Flow. The middle of the regulation can be set in **Percent** or **MW**. If the middle of the regulation range is set in **Percent**, the middle of the range is set to the specified percent of the present middle of the regulation range. If the middle of the regulation range is set in **MW**, then the middle of the regulation range is set to the specified amount. Regardless of the method used to set the middle of the regulation range, the regulation range will remain the same, but the Regulation Minimum MW Flow and Regulation Maximum MW Flow will be adjusted so that the specified middle of the regulation range is met.

Change By

The Change By action will change the middle of the regulation range of the selected phase shifter by the specified **Amount**. The middle of the regulation range is calculated by taking the average of the Regulation Minimum MW Flow and the Regulation Maximum MW Flow. The middle of the regulation range can be changed in **Percent** or **MW**. When the middle of the regulation range is changed in **Percent**, the change is based on the specified percent of the present middle of the regulation range. When the middle of the regulation range is changed in **MW**, the middle of the present regulation range is changed by the specified amount. Regardless of the method used to change the middle of the regulation range, the regulation range will remain the same, but the Regulation Minimum MW Flow and Regulation Maximum MW Flow will be adjusted so that the specified middle of the regulation range is met.

3-Winding Transformer

Open

The Open action will set the Status of the selected three-winding transformer to *Open* if the three-winding transformer Status is *Closed*. If the three-winding transformer Status is already *Open*, then this action does nothing.

Close

The Close action will set the Status of the selected three-winding transformer to *Closed* if the three-winding transformer Status is *Open*. If the three-winding transformer Status is already *Closed*, then this action does nothing.

Solve Power Flow

The solve power flow type is unique in that you can include an action that forces Simulator to solve the power flow as part of the contingency. There are rare special cases of sophisticated

contingency definitions this can be used for, as requested by one or more PowerWorld customers. The load flow is already generally solved for each contingency as part of the processing.

If a contingency has one or more Solve Power Flow actions, then there will be a few changes in how the contingency definition display behaves.

- Sorting of the list of actions is no longer allowed. This is because the order of the actions is now important to how the contingency is processed.
- When you right-click on the list of contingency actions there will be two new options for Move Up and Move Down. These can be used to reorder the actions.
- On the Contingency Definition Dialog, there will be up/down arrows on the right of the dialog that may be used to reorder the actions.

Contingency Block

The individual actions of the Contingency Block are applied according to how they are defined. There are no Action Types available for a Contingency Block.

Choose the Element

Use this portion of the dialog to choose the element involved in this action. This behaves the same as the [Advanced Find Dialogs](#) used throughout the software.

Action Type

Defines the change specified by the contingency action. The Action Types available depend on the **Element Type** selected. Possible Action Types include: Open, Close, Move, Set To, Change By, Bypass, and Inservice. The behavior of each of these actions for the different elements is described with the **Element Types**.

Amount

Enterable fields used to specify the quantity of change desired for the contingency element. The Amount fields are enabled when the **Action Type** is set to *Move*, *Set To*, or *Change By*. The availability of these Action Type options changes depending on the **Element Type** selected. The top field is used to define a *Constant*, *Field*, or [Model Expression](#) depending on the entry selected in the bottom drop-down box. Use the **Find** button to display a dialog listing *Fields* or *Model Expressions*. This dialog behaves the same as the [Advanced Find Dialogs](#) used throughout Simulator.

The Amount of change that occurs is based on the value of the top field entry. The top entry field is a constant, field name, or model expression name. When this entry is a field name, then the value of the change is based on the value of the selected element field when the contingency occurs. When this entry is a model expression, then the value of the change is based on the value of the selected model expression when the contingency occurs.

The Amount of change that occurs for the selected element is based on the value entered in the top field and the parameter set with the **in** options. The availability of the **in** options will change based on the Element Type selected and the Action Type selected. The availability of these options is described in detail with the **Element Types**.

Make-up Power Sources

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting real power imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information.

Model Criteria

Click the **Add** button to specify a criterion under which the contingency action will occur. Either a [Model Filter](#) or [Model Condition](#) can be selected for the Model Criteria. For example, the user can specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. This condition should be specified as a Model Condition and then this condition selected as the Model Criteria. Normally, no Model Criteria will be specified for a given action and this field will be blank.

Comment

An optional user-specified comment string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#) or with the case PWB file.

Make-Up Power Sources

Generator, load, injection group, and switched shunt contingencies may cause imbalances between generated power and demand. The default way to handle these imbalances is to assign them to the system slack. However, this is probably unrealistic and, depending on the size of the unit or load involved, convergence problems may result.

To implement more local compensation for generation, load, injection group, and switched shunt changes, use the Make-Up Power Sources Dialog. This dialog can be accessed by from the [Contingency Element Dialog](#) by clicking the button labeled **Make-Up Power Sources**. Most of the dialog is occupied by a grid that lists bus numbers and relative contributions. This grid is a [Case Information Display](#), so its behavior should be familiar. For example, right-click on the grid to display its local menu.

We shall call those buses that must compensate for the changes caused by a generator, load, injection group, or switched shunt contingency "compensators". To insert a new compensator, select **Insert** from the grid's local menu. This opens another dialog, where you should specify the bus number of the compensator and its contribution. Specify the contribution of the compensator either as a percentage or as a fixed number of MW. If a particular contingency has multiple compensators, then the choice of basis for the contribution (either MW or percent) should be consistent for each. After identifying the compensator, click **OK**. The grid should update with your newly added compensator.

To delete an existing compensator, select it from the grid and select **Delete** from the local menu.

To determine how much each compensator contributes to the imbalance, each contribution value is normalized to the total sum of all contribution values. This allows the values to be entered in either percent or MW. Keep in mind that when entering the values in percent that the sum of all contribution values should equal 100 or the actual contribution amounts will not occur in the expected percentages.

Compensators account for the changes caused by a generator, load, injection group, or switched shunt contingency by changing either generation or load to satisfy its defined contribution. For example, suppose compensator contributions are specified as percentages, and Simulator needs to compensate for 100 MW lost in a particular generation contingency. Suppose the contingency has 4 compensators defined as follows:

Bus Number Contribution

1	20
2	30
3	10
4	40

Suppose buses 1 and 2 are load buses, and buses 3 and 4 are generators. Then bus 1's load will decrease by 20 MW, bus 2's load will decrease by 30 MW, bus 3's generation will increase by 10 MW, and bus 4's generation will increase by 40 MW. These changes will be instituted regardless of the compensators' operating limits or AGC status. In fact, any compensating units will be set off AGC to ensure that the prevailing AGC control does not distort the dictates of the contingency. Furthermore, maximum MW limits on generators will not be checked.

If a compensator has both generators and loads attached to it, the generator will take precedence. The generator will function as the compensating device, and the load will be left unchanged.

Contingency Make-Up Sources Dialog

Simulator allows you to specify buses that will serve as compensators to account for changes in the balance between generation and demand caused by generator and load contingencies. This dialog allows you to define or adjust the amount of power supplied by a compensator. For more information, please see [Make-Up Power Sources](#).

Tutorial: Contingency Analysis

This tutorial will walk you through the basic commands necessary to insert contingencies and have Simulator automatically analyze the results. Please see [Introduction to Contingency Analysis](#) for the necessary background information regarding the capabilities and uses of the Contingency Analysis tool.

For this tutorial, we will use an existing 7-bus case.

- Open case B7SCOPF from the *Program Files/PowerWorld/Simulator/Sample Cases* directory.
- Ensure Simulator is in [Run Mode](#).
- Select **Contingency Analysis** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. Simulator opens the Contingency Analysis Dialog.

The next section of the tutorial discusses Defining Contingencies and provides an example of inserting a single element contingency.

1 2 3 4 5 6 7 8 9 10 11

Next

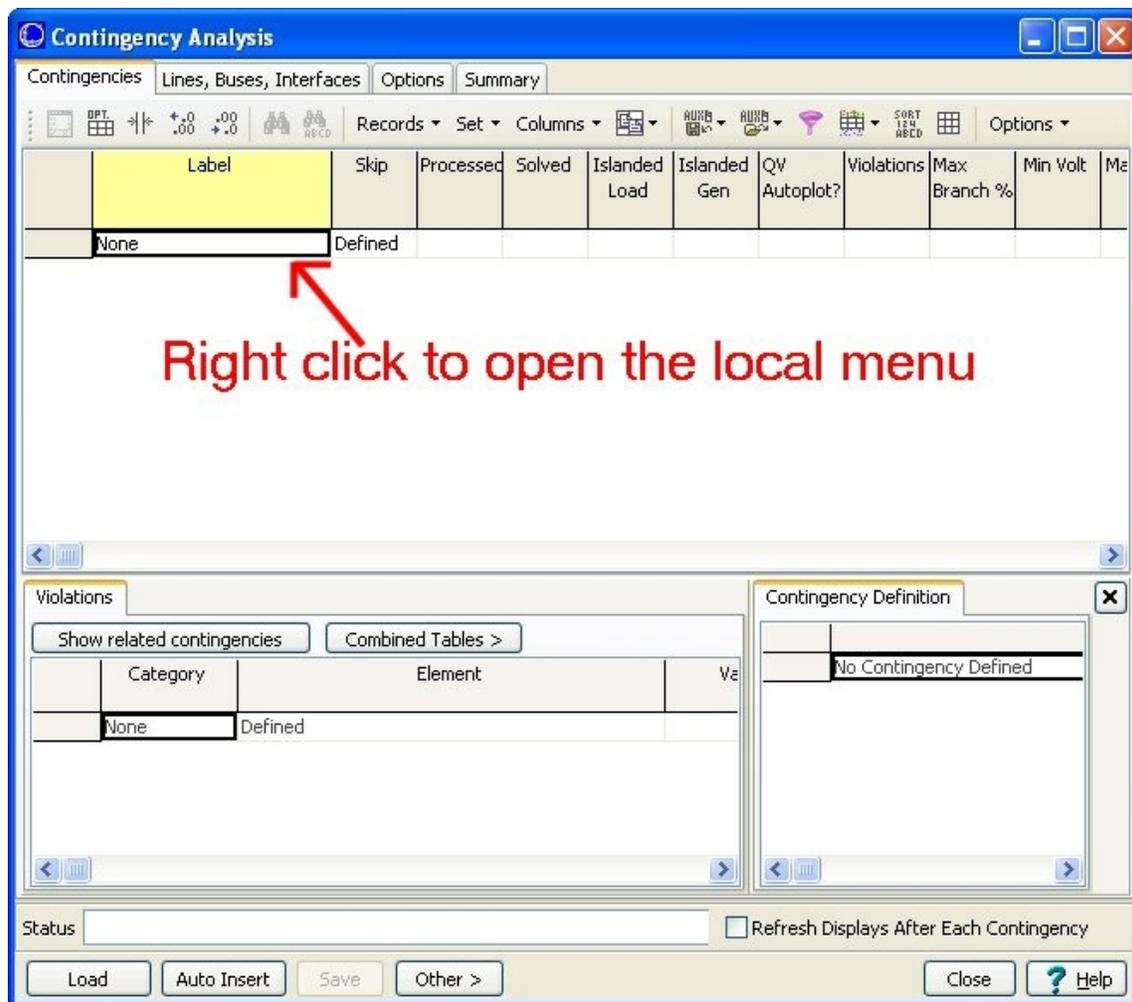
Tutorial: Contingency Analysis - Page 2

Defining Contingencies

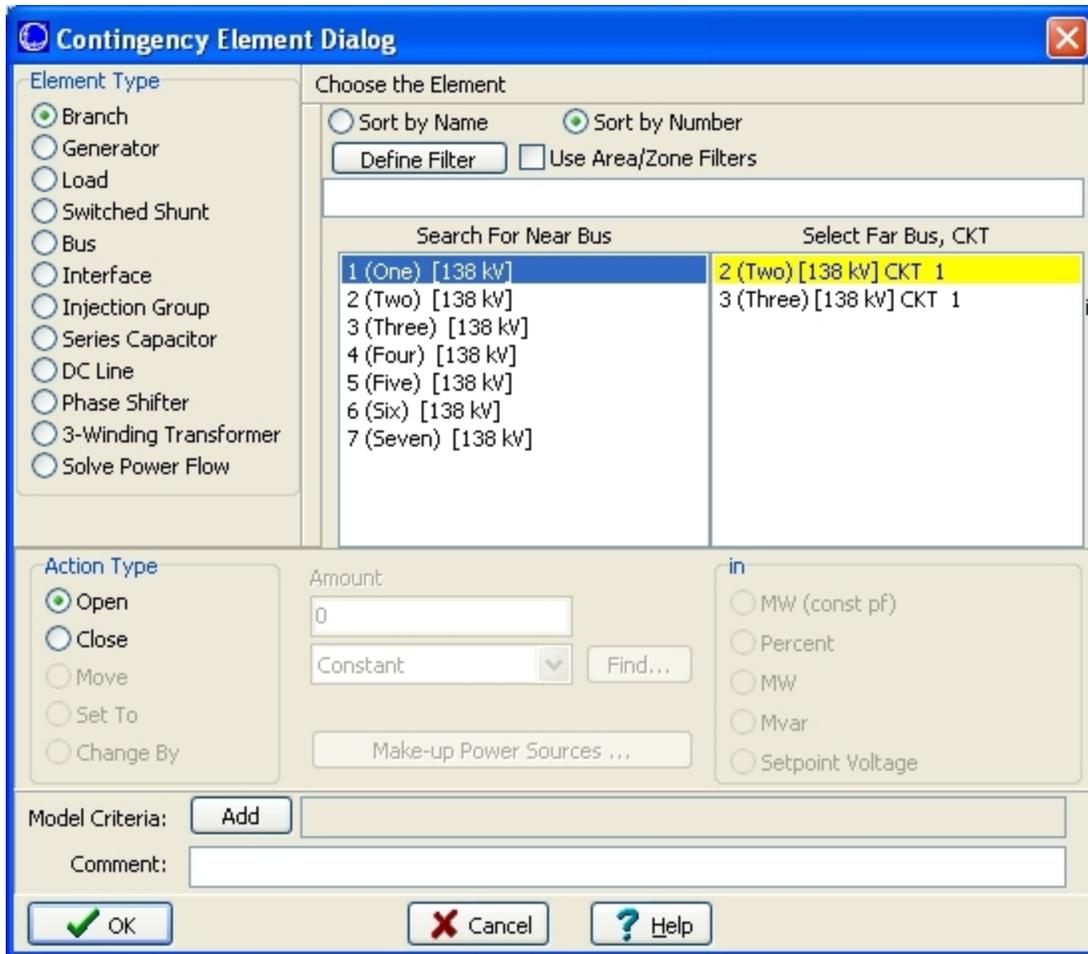
Contingencies can be either Single Element or Multiple Element. A single element contingency has only one associated contingent action. Click [here](#) for more information on the terminology used. There are four options for defining contingencies. The user may: [Load Contingencies from a File](#), [Auto Insert Contingencies](#), or use the local menu to either **Insert** contingencies or **Quick Insert of Single Element Contingency**. This tutorial will utilize the Auto Insert and Quick Insert tools.

Quick Insert of Single Contingency Element

- Right-click on the grid (as shown below) and select **Insert Special > Quick Insert of Single Element Contingency** from the local menu. Simulator opens the [Contingency Element Dialog](#).



Contingency Analysis Dialog



Contingency Element Dialog

Note: The active fields in this dialog depend upon which **Element Type** is selected. The defined action for this contingency will be to change the load at bus 2 by 25%.

- Select *Load* under **Element Type** and **2 (2) #1 [138 KV]** under **Choose the Element**.
- Select *Change By* under **Action Type**.
- Enter *25* for **Amount**, choose *Constant* from the drop-down box, then select *Percent* under the field labeled **in**.

Note that you can enter either positive or negative values in the Amount field to specify the direction that you want the change to occur (i.e. for a load contingency, entering a positive Amount increases the power consumed by the load, a negative value would decrease the load's power consumption).

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information. See Model Criteria and Comment in the [Contingency Element Dialog](#) for more information on those fields. For the purpose of this tutorial, we will not use these fields.

- Click **OK** to insert the contingency element.

The lower right portion of the Contingencies Tab contains a Contingency Definition list display. This display lists all elements (actions) associated with the selected contingency. Since we inserted a single element contingency, there is only one element shown in the Definition.

Contingency Definition				
	Actions	Model Criteria	Status	Comment
1	CHANGE LOAD AT BUS Two (2) BY 25.00 MW (cnst pf)		CHECK	

Contingencies Tab – Definition Display

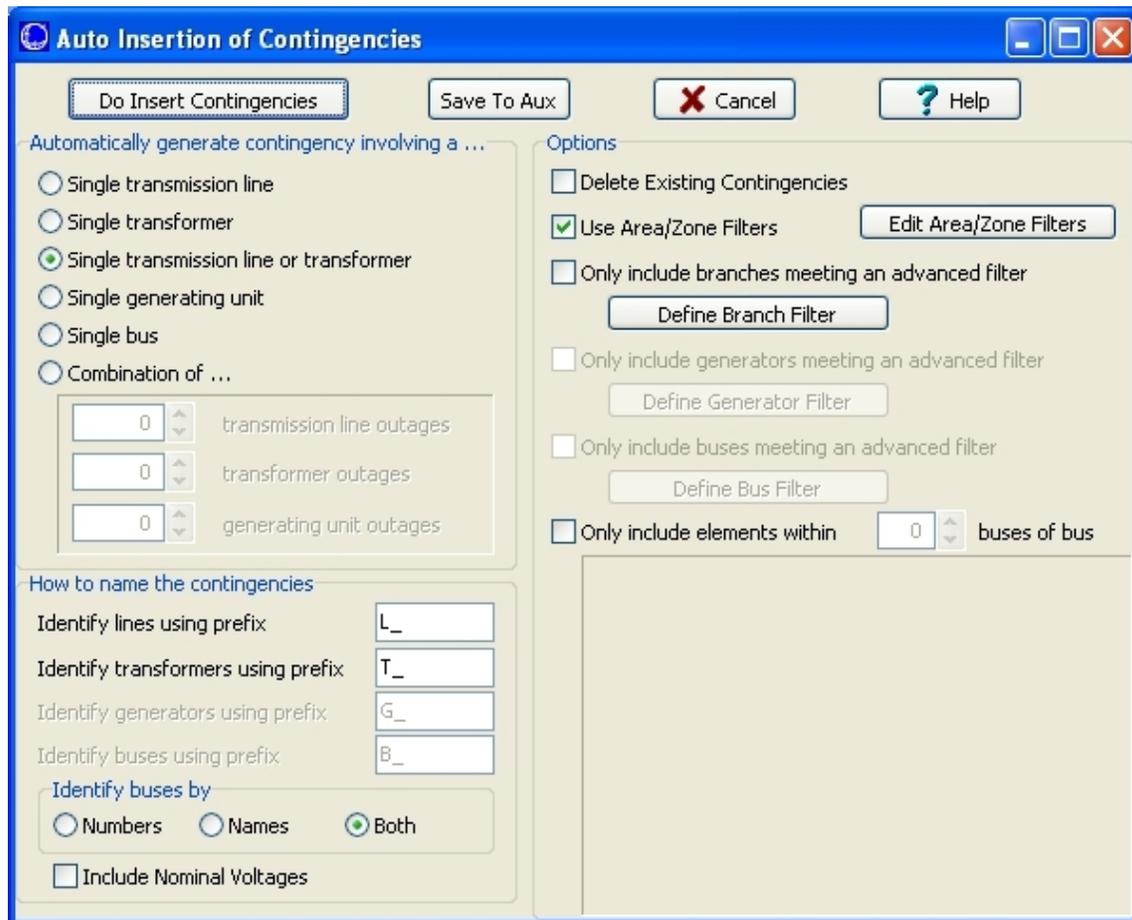
The next portion of the tutorial provides an example of Auto Insertion of Contingency Elements.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 3

Simulator allows you to automatically generate a contingency list containing branch, generator and/or bus outages. To accomplish this, right-click in the Contingencies grid on the [Contingency Analysis Dialog](#), and select **Insert Special > Auto Insert Contingencies** from the popup menu. This opens the [Auto Insertion of Contingencies Dialog](#).



Auto Insertion of Contingencies Dialog

We will insert contingencies for all branches and generators. This will require two executions of the auto insert tool.

- Verify that **Single Transmission Line or Transformer** is selected.

When using the Auto Insert tool, you can limit the contingencies inserted to only those meeting a defined filter. We want to insert contingencies for all branches and generators so no filtering is desired.

- Click to remove the checkmark in **Use Area/Zone Filters**.

- Verify that no other options are selected.
- Click the **Do Insert Contingencies** button to accept the remaining default values and automatically insert the branch contingencies.
- Click **Yes** when asked to confirm the insertion of 11 contingencies.

Note that the Contingencies Tab of the Contingency Analysis Dialog now shows 12 contingencies.

- Right-click on the list display of the contingencies tab and select **Insert Special > Auto Insert Contingencies** from the local menu.
- Select **Single Generating Unit** then click the **Do Insert Contingencies** button. Click **Yes** to complete the auto insert of 4 generator contingencies.

Note that the Auto Insert tool did not insert a contingency for the generator connected to the Slack Bus. You can manually insert slack bus generator contingencies.

The contingencies tab now shows 16 contingency records. You can click on an individual record and view its information in the [Contingency Definition](#) section of the Contingencies Tab.

The next section of the tutorial discusses running the contingency analysis.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 4

When running the Contingency Analysis, the user has three options:

1. Run every contingency on the list of contingencies (click **Start Run** on the Contingencies Tab, click **Start** on the Summary Tab or select **Run Contingency Analysis** from the local menu of the list of contingencies)
2. Run a single contingency (discussed in the advanced tutorial section) or
3. Run a single contingency then save the post-contingency state as the new reference state (also discussed in the advanced tutorial section)

We will run every Analysis in the list for this portion of the tutorial.

- Press Start Run on the Contingencies Tab. Note: **Pause** and **Abort** buttons are available on the dialog while the contingency analysis is running. These may prove useful when processing a long list of contingencies.

The results from the run are shown in the Contingency Analysis Dialog.

The screenshot shows the 'Contingency Analysis' dialog box with the 'Contingencies' tab selected. The main table displays the following data:

	Label	Skip	Processed	Solved	Islanded Load	Islanded Gen	QV Autoplot?	Violations	Max Branch	Min Volt	Max Volt	Max Interface %
1	L_00001One-00002TwoC1	NO	YES	YES			NO	1	149.3			
2	L_00007Seven-00005FiveC1	NO	YES	YES			NO	1	113.5			
3	L_00002Two-00006SixC1	NO	YES	YES			NO	1	103.9			
4	L_00002Two-00003ThreeC1	NO	YES	YES			NO	0				
5	L_00002Two-00005FiveC1	NO	YES	YES			NO	0				
6	L_00003Three-00004FourC1	NO	YES	YES			NO	0				
7	L_00004Four-00005FiveC1	NO	YES	YES			NO	0				
8	L_00002Two-00004FourC1	NO	YES	YES			NO	0				
9	L_00006Six-00007SevenC1	NO	YES	YES			NO	0				
10	L_00006Six-00007SevenC2	NO	YES	YES			NO	0				
11	L_00001One-00003ThreeC1	NO	YES	YES			NO	0				

Below the table, the 'Violations' section shows a summary table:

Category	Element	Val
1 Branch Amp	One (1) -> Three (3) CKT 1 at One	406.00

The 'Contingency Definition' section shows:

Acti
1 OPEN Branch One 138.0 (1) TO Tw

The status bar at the bottom indicates: 'Finished with 3 Violations and 0 Unsolvable Contingencies. Initial State Restored.' There are buttons for 'Load', 'Auto Insert', 'Save', 'Other >', 'Start Run', 'Close', and 'Help'.

Contingency Analysis Dialog – Contingencies Tab

Note: The **Refresh Displays after Each Contingency** option (lower right) can slow down the analysis significantly when running a long list of contingencies.

The contingency analysis results are sorted on the contingencies tab in descending order by worst violation. We see that there were three violations resulting from the contingency analysis (one for each of the first three contingency records displayed) and that no unsolvable load flows resulted (as shown in the Status Section of the Contingencies Tab.) If you enlarge the Contingency Analysis Dialog (by dragging either side or the corner of the pane) you see more information in the Status Window, specifically, "Finished with 3 Violations and 0 Unsolvable Contingencies. Initial State Restored."

Contingency Analysis always stores a Reference State or pre-contingency state. The Reference State stores information pertaining to: buses, switched shunts, limit groups, loads, branches, generators, areas / super areas, and power flow solution options.

Both prior to and following completion of solving a list of contingencies, the reference state is loaded into memory. This ensures that all contingency analysis solutions start from a common base case and that the system is restored to its initial state following a solution. The last sentence in the Status portion of the dialog, "Initial State Restored," tells the user that the simulation case was restored to the reference state upon completion of the Contingency Analysis Run. See [Contingency Case References](#) for more information on the Reference State.

The [Violations Section](#) (lower left) of the Contingencies Tab provides a description of each violation resulting from the execution of the contingency selected in the list of contingencies. Scroll through the list of contingencies to view information about the resulting violations for each.

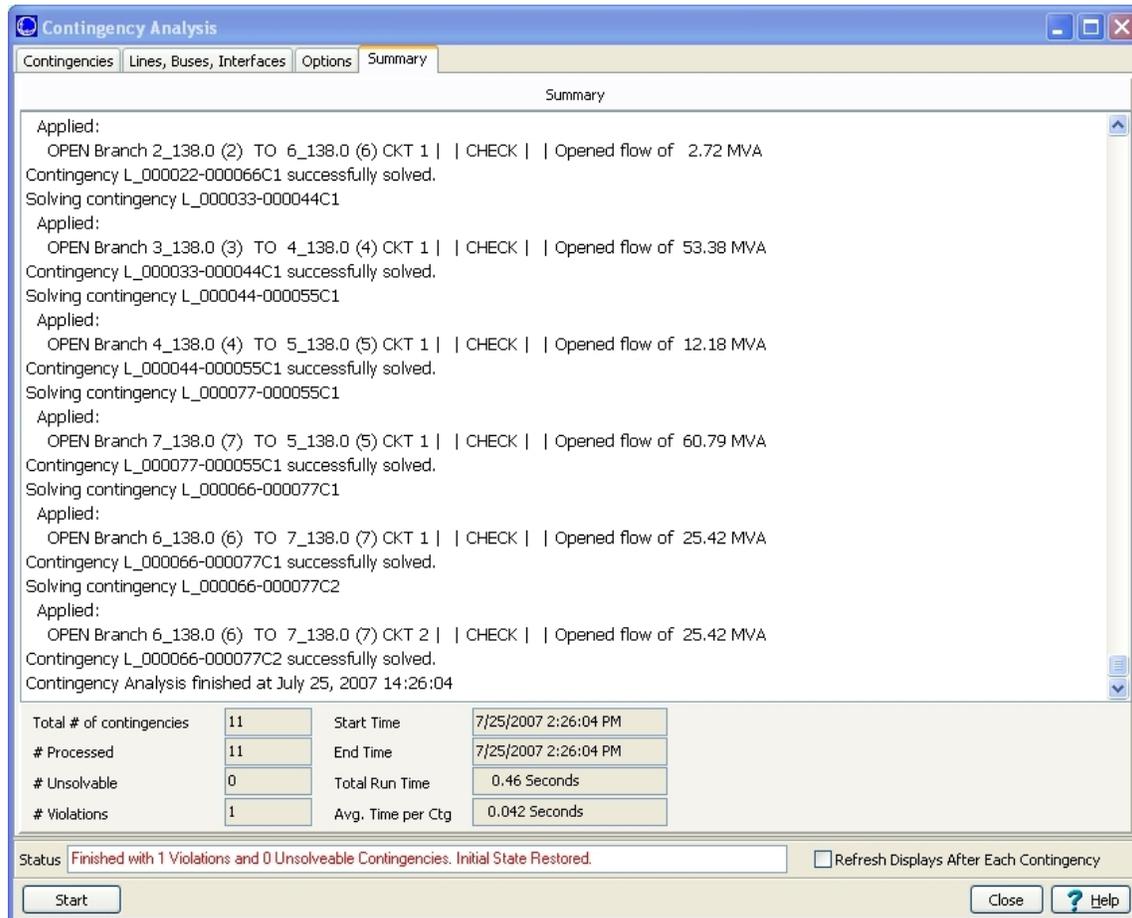
Note: You can hide the Contingency Definition section of the Contingencies Tab by clicking the  button.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 5

The [Summary Tab](#) of the Contingency Analysis Dialog provides the status of the present contingency analysis run.



Contingency Analysis Dialog – Summary Tab

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 6

- Click on the Lines, Buses, Interfaces Tab of the dialog.

The Lines, Buses and Interfaces Tab contains four sub-tabs: Lines/Transformers, Buses, Interfaces, and Nomogram Interfaces. The information contained on each of the sub-tabs provides an alternate method of viewing information similar to that contained on the [Contingencies Tab](#). The individual tabbed sheets show all model objects defined in the case (subject to area/zone/owner filters) whether each is associated with a specific contingency or not.

The screenshot shows the 'Contingency Analysis' dialog box with the 'Lines, Buses, Interfaces' tab selected. The 'Lines/Transformers' sub-tab is active, displaying a table with 11 rows of data. Below this, there are two smaller tables: 'Contingencies' and 'Contingency Definition'.

	From Number	From Name	To Number	To Name	Circuit	Xfrmr	Aggr Percent Overload	Violations	Max % Loading Cont.
1	1 1		3 3		1	No	31.09	1	131.09
2	1 1		2 2		1	No	28.71	1	128.71
3	2 2		5 5		1	No	6.30	1	106.30
4	2 2		3 3		1	No	0.00	0	
5	2 2		6 6		1	No	0.00	0	
6	3 3		4 4		1	No	0.00	0	
7	6 6		7 7		1	No	0.00	0	
8	7 7		5 5		1	No	0.00	0	
9	4 4		5 5		1	No	0.00	0	
10	6 6		7 7		2	No	0.00	0	
11	2 2		4 4		1	No	0.00	0	

	RANK Overload	Label	Category	Val	Lim
1	2.2	L_000011-000022C1	Branch MVA	157.31	

	Actions
1	OPEN Branch 1_138.0 (1) TO 2_138.0 (2) C

Status: **Initialized** Refresh Displays After Each Contingency

Buttons: Load, Auto Insert, Save, Other >, Start Run, Close, Help

Contingency Analysis Dialog – Lines, Buses, Interfaces Tab

The user can select any model object on its respective sheet to see how many times a violation occurred on the device during the contingency analysis. When a particular device is selected that had at least one violation during the contingency run, the [Contingencies](#) and [Contingency Definition](#) sections give the details of the specific contingencies that caused the violation (or violations) on the selected device. For example, on the Lines/Transformers sub-tab:

- Select the line from 2 to 5 as shown above. This line had one violation on the most recent contingency analysis run. The contingencies section shows which contingency caused the violation and the contingency definition section details the elements that define the contingency.
- Switch back to the Contingencies Tab of the dialog.
- Scroll down in the list of contingencies and select the contingency labeled CHANGE LOAD AT BUS 2 (2)
- Right-click on the element displayed in the Contingency Definition Section of the Contingencies Tab
- Select **Show Dialog** from the local menu.
- Modify the element's **Action** to: *Set To 600 MW (const pf)* using the **Action Type**, **Amount**, and **in** fields of the Contingency Element Dialog.
- Click **OK** to close the dialog. The Contingency Definition should now show SET LOAD AT BUS 2_138 (2) TO 600.00 MW (const pf).
- Click **Start Run** on the dialog and **Yes** when asked to confirm.
- Following the run, the Status field now shows, "Finished with 7 Violations and..." The increase in load resulted in four contingency violations not present during the last run.
- Again select the contingency labeled SET LOAD AT BUS 2... on the list display. The Violations section shows the four branch violations that occurred for the selected contingency.
- Now switch to the Lines, Buses, Interfaces tab.
- Select the line from 2 to 5. This line experienced violations under two contingencies during the run. The Contingencies section now shows the details of both contingencies (opening line 5 to 7 and changing the load at bus 2) that caused overloads on the line. Notice that the information in the Contingency Definition section is specific to the Contingency selected in the Contingencies section.

Note: The **Show Related Contingencies** and **Show Other Violations** buttons (on the Contingencies and Lines, Buses, Interfaces tabs respectively) provide a fast method of switching between the two tabs and viewing related information.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 7

You now have enough information to effectively utilize PowerWorld Simulator's Contingency Analysis tool. The remaining portions of the tutorial introduce the full capabilities of the Contingency Analysis tool and provide links to the applicable Help files.

Contingency Records

There are four methods of defining contingencies. See [Defining Contingencies](#) for more information. Contingency Records can also be [saved to](#) or [loaded from](#) a file.

Contingency Blocks and Global Actions

The user may desire to have a common set of actions occur during more than one (or all) contingencies. Instead of repeatedly defining the same contingency element (or elements) in multiple contingencies, Simulator provides the option to use Contingency Blocks and/or Global Actions.

A [Contingency Block](#) is a set of contingency actions that can be defined and then called upon by individual contingency records.

[Global Actions](#) are actions defined by the user that will occur during all contingencies.

Contingency Solution Options

By default, the contingency analysis will use the same options as the power flow algorithm when solving each contingency. You may also override these options for all contingencies, and/or for a specific contingency. This results in the ability to set the power flow solution options in contingency analysis at three different levels

1. Contingency Specific Options (see [Contingency Definition Dialog](#))
2. Contingency Analysis Options (see [Contingency Options Tab](#))
3. General Power Flow Solution Options (see [Power Flow Solution Options](#))

When Simulator executes a particular contingency, it will first look at options specified for that contingency. Any options defined for the contingency will be used. Other options set to *use default* will look to the Contingency Analysis Options. Again, any options defined for contingency analysis will be used. Finally, options marked in the Contingency Analysis Options as *use default* will be set to the same setting as the power flow solution options.

[1](#) [2](#) [3](#) [4](#) [5](#) [6](#) [7](#) [8](#) [9](#) [10](#) [11](#)

[Prev](#) [Next](#)

Tutorial: Contingency Analysis - Page 8

Reference State

The [Reference State](#) (introduced on page 4 of the tutorial) can prove very useful to advanced users if they know how to exploit its capabilities. When running a list of contingencies the reference state is loaded into memory prior to executing each contingency. The system is restored to the reference state following the contingency analysis run. In addition to running an entire contingency list, the user also has the option to solve individual contingencies by right clicking on the desired contingency - on the Contingencies tab of the Contingency Analysis Dialog - and selecting either **Solve Selected Contingency** or **Solve and Set as Reference** from the local menu.

Solve Selected Contingency causes Simulator to first load the reference state into memory then solve the contingency. THE SYSTEM STATE IS NOT RESTORED TO THE REFERENCE STATE FOLLOWING THE SOLUTION; the system state then reflects the power system flows of the post-contingency state. The advantage of this approach is the ability to implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by the contingency. Be aware; however, that prior to solving another contingency, Simulator will reset the system state to the reference state thereby removing all modifications made following the previous contingency solution. The user may also automatically restore the system state to the reference state by selecting **Other > Restore Reference** from the Contingency Analysis Dialog.

Solve and Set As Reference acts the same as Solve Selected Contingency with one exception. After executing the contingency, the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use the post-contingent state as the Reference State.

Make-up Power Sources

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 9

Contingency Elements

A contingency element consists of a single Contingency Action and its associated Model Criteria, Status and Comment (optional). Multiple Contingency Elements can be defined for a single Contingency.

Contingency Action - Click [here](#) for a list of available contingency actions.

Model Criteria - Model Criteria are criteria under which the contingency action will occur and consist of both [Model Conditions](#) and [Model Filters](#).

Status – The status field of a contingency element can take one of four values: CHECK, ALWAYS, NEVER or POSTCHECK.

- CHECK - The action will be executed if the Model Criteria are true or if no Model Criteria are specified. CHECK is the default status setting.
- ALWAYS - The action will always be executed, regardless of the Model Criteria.
- NEVER - The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- POSTCHECK – Following completion of all CHECK and ALWAYS actions, the contingency analysis tool runs the load flow solution. POSTCHECK actions are then addressed recursively until all are complete. The execution of POSTCHECK actions follows the same rules as CHECK actions with the exception of not being checked until the load flow has been solved. If the Model Criteria specified for the POSTCHECK action are met in the solved load flow solution (or if no Model Criteria are specified), then the action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped.

For more information, see [Contingency Definition Display](#).

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 10

Contingency Analysis Dialog – Options Tab Overview

The bulk of the [Options Tab](#) contains five sub-tabs, each of which concerns a different aspect of the contingency analysis. A brief description of the actions allowed via each sub-tab follows. Click on the headings below for more information on each.

Modeling Sub-tab

The modeling tab allows the user to:

- Define the Contingency Analysis Load Flow Calculation Method
- Specify Limit Monitoring Settings
- Instruct Simulator to retry the solution using the Robust Solution Process following a failure to converge
- Specify the use of specific solution options for contingencies
- Specify Make-up Power for the post-contingency solution

[Advanced Limit Monitoring Sub-tab](#)

The Advanced Limit Monitoring Tab allows you to shape how limit violations are detected and reported.

[Advanced Modeling Sub-tab](#)

The Advanced Modeling tab allows the user to:

- Define Contingency Blocks and Global Actions (refer to [Page 7](#) of the tutorial)
- Define Model Criteria (Expressions, Conditions and Filters)
- Specify Model Criteria Options
- Define Bus Load Throw-Over Records – BLTRs provide the capability to define how load at a bus should be transferred to a different bus if the original terminal bus becomes disconnected from the system during a contingency
- Define Generator Maximum MW Responses in Post-Contingency – allows the user to limit the absolute MW response of a generator during a contingency
- Define Generator Line Drop Compensation (LDC) and Reactive Current Compensation (RCC) – LDC and RCC controls allow the user to model the real-time control done at some real generators
- Specify a post-contingency aux file to be loaded at the start of each contingency. This allows the setting of very specialized post contingency settings.
- Specify how reactive power is modeled for DC calculation methods

[Report Writing Sub-tab](#)

Simulator can produce a report that details the results of the contingency. The Report Writing Tab allows you to control the content and appearance of the report.

[Miscellaneous Sub-tab](#)

The Miscellaneous sub-tab provides options pertaining to the loading and saving of contingency records as well as specifying how the reference state should be established when the contingency analysis tool is accessed.

[1](#) [2](#) [3](#) [4](#) [5](#) [6](#) [7](#) [8](#) [9](#) [10](#) [11](#)

[Prev](#) [Next](#)

Tutorial: Contingency Analysis - Page 11

Contingency Analysis Dialog – Other Contingency Actions

The **Other >** button on the [Contingencies Tab](#) and the [Lines, Buses, Interfaces Tab](#) of the [Contingency Analysis Dialog](#), provides access to a number of additional contingency actions. Some of the actions available include:

- Deleting all contingencies
- Clearing contingency results
- Setting or restoring the reference state
- Producing combined tables of results
- Producing detailed reports of results
- Comparing lists of contingencies
- Filtering results
- Auto-filling blank comment fields and performing sensitivity calculations such as OTDFs and PTDFs.

For more information, see [Other Contingency Actions](#).

1 2 3 4 5 6 7 8 9 10 11

Prev

Fault Analysis

Fault analysis can only be performed when Simulator is in [Run Mode](#). There are three ways to start a fault analysis study:

- Go to the [Tools](#) ribbon tab, and choose **Fault Analysis** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).
- Right click on a bus and choose **Fault...** to perform a fault analysis at that bus
- Right click on a line and choose **Fault...** to perform a fault analysis at that point on the line

All of these options will open the [Fault Analysis](#) dialog. If you opened the dialog by right-clicking on a bus or line, the fault information on that bus or line will already be filled in. If you selected the **Fault Analysis...** option from the [Tools](#) ribbon tab, the information about the location of the fault will need to be provided.

Fault Analysis Dialog

The **Fault Analysis** dialog can be used to perform a fault analysis study on the currently loaded power system. A fault study can only be performed while Simulator is in [run mode](#), since the load flow must be validated and solved before a fault study can be calculated. If you are observing fault analysis results in the fault analysis dialog and switch to [edit mode](#), the dialog will automatically be closed and the fault analysis results will be cleared from memory.

Simulator stores fault data in the PowerWorld binary file along with the load flow data, but by default most other load flow formats store fault data in separate files. The fault data can be stored in and loaded from an external file, but if no fault data is present in a PowerWorld binary file or loaded from an external file before a fault analysis is run, Simulator will use the load flow data as default values for the analysis. Fault data values can also be modified for specific devices by opening a specific device's information dialog and looking at the **Fault...** tab. Devices that require sequence specific data for fault analysis are [buses](#) (for sequence load injections), [generators](#), [switched shunts](#), [transmission lines](#), and [transformers](#).

Phase shifts in a fault analysis calculation can be very important for calculating the correct fault currents and voltages throughout the system. The phase shifts that are applied for transmission lines and transformers are taken from the load flow values of phase entered with each specific transmission element. While transformers can have their transformer configurations specified (i.e. Delta-Wye, Grounded Wye-Delta, etc.), these configurations are **NOT** used to determine phase shift angles, **ONLY** to determine the proper grounding on each side of the transformer. The phase shifts that are applied are taken from the load flow data phase values for the transmission elements. If no phase shifts are entered in the load flow data, the fault analysis will treat all elements as having zero phase shift. Phase shift values can be entered manually for each transmission element, but are also included in most load flow formats and will be read into Simulator when loading a load flow data file.

Note that the bus chosen for the fault is always set to a 0 degree reference, and all other buses are shifted according to this reference.

Fault Data

The **Fault Data** tab is where the type and location of the fault are specified, and where the results of the fault analysis can be seen in tabular format.

Fault Location

Choose to perform the fault at a bus location, or at a point somewhere on a line. If Bus Fault is selected, the only information needed is the bus number, which needs to be entered in the Fault Bus field. If an in-line fault is desired, the from and to bus numbers, circuit ID, and location of the fault (entered in percent of total line length, measured from the From Bus) will need to be given. Selecting the **Fault...** option from the bus or line local menus will automatically set up the Fault Location fields.

Fault Type

Choose from one of four types of fault to calculate at the fault location:

Single Line - to - Ground Computes a single phase line - to - ground fault using a user defined ground fault impedance. The phase evaluated is always referenced as phase A.

Line - to - Line Computes a line - to - line fault, assuming an impedance of $999 + j999$ to ground. Phases B and C are always referenced as the faulted phases.

3 Phase Balanced Balanced three-phase line fault - to - ground using a user-defined ground fault impedance.

Double Line - to - Ground Computes a line - to - line - to - ground fault, using a user defined ground fault impedance.

Current Units

Allows you to choose to observe the fault currents in per unit current or actual Amps.

Online Display

The string grids at the bottom of the **Fault Analysis** dialog will display the fault results in tabular format, but the results can also be viewed graphically on the online diagram by selecting an option from this group of options. Any of the three phase values can be selected for display individually, or all three phases can be viewed simultaneously. Viewing all three phases of information simultaneously can result in an abundance of information on the diagram at one time, so selective placement of the necessary bus and line fields may need to be considered when planning on viewing the fault analysis results graphically on the online diagram.

The fields necessary on the online diagram for display of the fault analysis results are:

- **Bus Voltage** and **Bus Angle** fields need to be present for a bus. When choosing to view fault analysis results, these two fields will be identified, and the actual load flow values will be replaced by the fault phase voltages (in per unit) and angles (in degrees).
- **AC Line MW Flow** and **AC Line Mvar Flow** fields need to be present for a line. These two fields will be identified, and the MW and Mvar values will be replaced by the fault phase current magnitudes (in Amps or per unit) and angles (in degrees).
- **Gen MW Output** and **Gen Mvar Output** fields need to be present for a generator. These two fields will be identified, and the MW and Mvar values will be replaced by the generator terminal fault phase current magnitudes (in Amps or per unit) and angles (in degrees).

Fault Current

Displays the magnitude and angle of the current at the fault location during the fault.

Calculate

Pressing this button will run the fault analysis. In order for the results to be calculated, the power flow has to be in a solved state for the results to have any relevance. Therefore the first thing performed when **Calculate** is pressed is to solve the power flow. You can observe this by viewing the [Message Log](#) when you run the calculation. Once the power flow has been solved, then the fault analysis calculations are run and the results displayed.

Clear

Pressing **Clear** will clear any fault analysis results currently in memory and displayed on the dialog.

There are also five informational displays at the bottom of this dialog for showing the fault analysis calculation results:

[Buses](#)

[Lines](#)

[Generators](#)

[Loads](#)

[Switched Shunts](#)

Fault Options

The **Fault Options** tab is where an impedance to ground can be defined at the fault location, where fault data can be loaded from or saved to an external file, and zero-sequence mutual impedances can be viewed or changed.

Fault Impedance

For any of the fault types calculated, a **Fault Impedance** can be included. A Resistance and Reactance can be entered as the path to ground of the fault, and is taken into account when calculating the fault current used to determine the rest of the fault values.

Load Data / Save Data

These two buttons allow loading from and saving to external files. Currently the two types of files supported are PSS/E Sequence Data files (.seq) and [PowerWorld Simulator Auxiliary files](#) (.aux). Either one of these formats can be loaded and saved.

Note that typically Simulator assumes that if no zero sequence data is given for a branch that the zero sequence impedance is defaulted to 2.5 times the positive sequence impedance. However, in the case of reading in data from a PSS/E sequence data file, you are given the option to instead treat branches with now explicitly given zero sequence data as open circuits in the zero sequence network.

Zero Sequence Mutual Impedances

Zero sequence mutual impedances can be stored and modified in the [Mutual Impedance Records](#) table. Usually the zero sequence mutual impedance parameters are read in from a sequence data file. However, it is also possible to insert and delete mutual impedance records from this table by right-clicking in the table and selecting **Insert** or **Delete** from the [local menu](#). When **Insert** or **Show Dialog** are chosen from the mutual impedance table local menu, the [Mutual Impedance Record dialog](#) will open, from which a mutual impedance record can be inserted or modified.

Pre-Fault Profile

Changing this option determines the pre-fault voltage profile to be used for the fault analysis calculations. The pre-fault profile selection affects the sequence Y-bus values, fault currents, and post-fault voltages.

Profile Options

Additional pre-fault profile options are available when the pre-fault profile selected is either Flat IEC-909 or Flat Classical.

XF Turns Ratios Set to 1 If checked, all transformer tap ratios are assumed at their nominal tap position.

Line Charging Set to 0 If checked, line charging capacitance is ignored in all calculations.

Shunt Elements Shunt elements (bus and line shunts) can optionally be treated normally, ignored in the positive sequence only, or ignored in all sequences.

IEC Parameters This option only applies to the Flat IEC-909 pre-fault profile. The pre-fault voltage magnitude can be specified for each bus. In addition, a generator power factor angle (in degrees) can be specified for use when generator currents need to be calculated based on bus voltage and power (real and reactive) delivered by the generator.

Matrices

The **Matrices** tab is where the positive, negative and zero sequence admittance matrices can be viewed for the fault. This tab is only visible when a fault has been calculated. The three pages on the Matrices tab each have the same functionality as other [Case Information](#) displays. The purpose of each display is to show the admittance matrix for the specified sequence. One of the most important features of these matrix displays is the ability to right-click on the display to bring up additional display options in the [local menu](#). Perhaps one of the most important options on the local menu is the ability to export the Y-bus admittance matrix to a Matlab M file, which allows import of the matrix into Matlab for additional manipulation, such as inverting the matrix to get the equivalent sequence Z-bus matrix.

Fault Analysis Bus Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase voltage magnitudes and angles are displayed. In addition, the sequence voltages and angles can also be added by modifying the display using the [Display/Column Options](#) dialog.

Fault Analysis Generator Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the generator. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the generator and into the terminal bus.

Fault Analysis Line Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for each end of the branch. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of or away from a bus.

Mutual Impedance Records

The **Mutual Impedance Records** table is a [Case Information Display](#) and can be customized like any other case information display. The zero sequence mutual impedance records displayed in this table can be either read from a sequence data file, or created manually by choosing **Insert...** from the local menu.

The common fields displayed on the Mutual Impedance Records display are:

L1 From Bus, L1 To Bus, and L1 Ckt ID

These fields represent the from bus number, to bus number, and circuit identifier for the first mutually coupled line.

L2 From Bus, L2 To Bus, and L2 Ckt ID

These fields represent the from bus number, to bus number, and circuit identifier for the second mutually coupled line.

Mutual R, Mutual X

The mutual impedance, in terms of the resistance and reactance (per unit). The dot convention of the mutual impedance assumes the From bus of each line to be the dotted terminal, with the sign of the mutual impedance values being set according to this convention.

L1 Mut. Start, L1 Mut. End

The starting point and ending point of the mutually coupled portion of the first mutually coupled line. The values are between 0 and 1, and represent a position on the line as a percentage of the total line length. These fields are only used when evaluating an in-line fault to determine the affect of the mutual impedance on each side of the fault point on the line.

L2 Mut. Start, L2 Mut. End

The starting point and ending point of the mutually coupled portion of the second mutually coupled line. The values are between 0 and 1, and represent a position on the line as a percentage of the total line length. These fields are only used when evaluating an in-line fault to determine the affect of the mutual impedance on each side of the fault point on the line.

Mutual Impedance Record Dialog

The **Mutual Impedance Record** dialog can be used to modify or add zero sequence mutual impedance records to the sequence data for a case. When the dialog is opened using the **Show Dialog...** option from the [Mutual Impedance Records](#) table local menu, the information for the record selected in the table will automatically be displayed. The information for that record can be modified, or a different record can be selected by selecting different lines in the Line 1 and Line 2 Identifier sections. Note that the drop down list of buses for the From Bus fields always contain all the buses in the case. However, once the From Bus has been selected, the drop down list of the corresponding To Bus field will only contain bus numbers of buses that are connected to the From Bus. If a mutual impedance record already exists for the lines selected, the information for that record will be displayed. If a mutual impedance record does not exist for the selected lines, then the mutual impedance fields will display default values. When the default values are changed, and either Save or OK are selected, a new mutual impedance record is added to the data.

Fault Analysis Load Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the load. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the bus and into the load.

Fault Analysis Switched Shunt Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the switched shunt. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the bus and into the switched shunt.

Contouring

Simulator can create and animate a contour map of various system quantities, such as voltage magnitudes and angles, MW transactions, transmission loading, and real and reactive load. Such displays resemble a contour map of temperatures like you might see shown on a weather forecast. Contouring can significantly improve understanding of a large interconnected system, helping identify congestion pockets and Mvar-deficient regions and providing an overview of how power flows through the bulk power system.

The [Contour Options Dialog](#) controls Simulator's contouring capabilities. To access it, click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting [local menu](#), or choose **Contouring** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab.

Contouring is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically removes the contour from the oneline diagrams. You can still choose to reapply the contour while in edit mode however.

Contouring Options

The Contour Options Dialog allows you to draw contour maps of many system quantities, such as bus voltages or angles, transmission line and interface MVA loadings, and transmission line and interface [PTDFs](#).

To access this dialog, click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting [local menu](#), or choose **Contouring > Contouring** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab. The Contour Options Dialog has three tabs: the [Contour Type](#) Tab, the [Contour Type Options](#) Tab, and the [Custom Color Map](#) Tab.

Contour Type

Object

Simulator can contour several different values. To specify what Simulator should contour, first choose the type of display object; the options are Bus, Line, Interface, Area, Generator, Substation, Zone or Injection Group. Once the object is chosen, you must then choose the numerical value to be contoured by specifying it in the **Value** dropdown box.

Note: to contour a value for a type of object, representations of that type of object must be present on the oneline diagram. Choosing to contour an object type that is not represented on the diagram will result in no contour being drawn on the diagram.

Value

Select the quantity to contour from the Value dropdown box or click the **Find...** button to find the desired field (it is normally easier to use the Find button). See NOTE at the end of this help topic.

Filter Object Values

The name of the advanced filter that is currently applied to the contour. Click Define to create a new [Advanced Filter](#) to use during contouring. Applying a filter means that *only* objects that meet the filter will affect the contour image. Objects which do not meet the filter will be ignored.

Data Points per Line

This specifies the number of data points used to represent the graphical line. The contouring algorithm is based on values at specific coordinates in the oneline space. When contouring transmission line or interface objects, the line is represented by a series of points spaced equally along the graphical line.

No Data Color

This setting allows you to choose the contour color used for parts of the contour for which there are not objects or nor data for the type of contour selected. The choices for No Data Color are Specific Color, Color Map Percentage, and Background Color. By default, Specific Color is selected and set to white. If you wish to change the specific color to use, click on the color box to the right and choose a different color from the popup dialog. If you select Color Map Percentage, the Color Map % field will become enabled, and you can select a value from 0 to 100. The value you enter will associate the No Data Color with the color located at that percentage in the selected color map. Lastly, if Background Color is chosen, the No Data Color will always be whatever color has been set as the normal background color for the oneline diagram.

Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values. You can also give the color key a title, unit label, and specify the number of digits to display in numerical values.

Title

Title for the color key.

Entry Labels

Units of the contoured value displayed on the color key.

Dec. Pts.

Number of decimal places of the contoured value displayed on the color key.

Scalar

Multiplication factor that can be applied to the values when drawing the color key. Normally this value should be 1.0.

Use Equal Spacing For Discrete Maps

This option will draw the color key with equal spacing for all colors in the map, regardless of how close or distant the values the colors represent. This option only applies to discrete color maps.

Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the [Custom Color Map](#) Tab.

Reverse Color Map Colors

Check this check-box to reverse the colors of the selected color map, so the low color becomes the high color, and vice versa.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Use absolute value

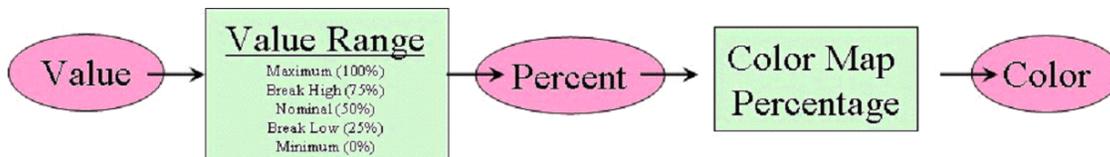
Check this check-box to use the absolute values of the quantity selected at the Value dropdown box (above).

Ignore Above Max

Check this check-box to completely ignore values above the maximum percentage. This means that data which is larger than the Max % will not be used in calculating the contour image. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so.

Values

For all color maps that come with PowerWorld by default, the conversion of a value to a color is done in two steps. The value is first interpolated to a percentage by using up to 5 user-specified values (maximum, break high, nominal, break low and minimum) that correspond to specific percentages (100%, 75%, 50%, 25%, and 0%). This percentage is then interpolated through the color map and a color is determined.



User may define their own color maps which convert percentage to color, but they may also define color maps which directly translate values into colors without the intermediate calculation of a percent. For more information on this see the creation of a [Custom Color Map](#).

The value range values are described as follows:

Maximum The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Break High This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

Nominal This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

Break Low This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

Minimum The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Note: a representation of the color map is shown to the right of the values.

Ignore Below Min

Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so.

Ignore Zero Values

Check this box to completely ignore zero values in the contour. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so. If there is a need to ignore values that are almost but not exactly zero, then create an Advanced Filter to Filter Object Values.

Interpretation

This combo box specifies how to interpret the values of the data points. The options are:

Fixed Values The data point values are not modified. The maximum, minimum, nominal, and break values are the ones entered directly in the units of the value being contoured. This is what should normally be used.

Dynamic Values The data point values are not modified. However the maximum, minimum, nominal, and break values are determined dynamically from the data point values as follows: Maximum = Maximum data point value; Minimum = Minimum data point value; Nominal = Average data point value; Break High = $(\text{Max} + \text{Average})/2$; and Break Low = $(\text{Min} + \text{Average})/2$.

Standard Deviations All the data point values will be used to determine a mean and standard deviation. The data point values will then be converted to represent the number of standard deviations they are from the mean. Thus a value equal to the mean will be changed to a 0, a value 1.5 standard deviations higher than the mean will be changed to 1.5, and so on.

Percentiles All the data point values will be sorted from lowest to highest. The value will then be set equal to the 100 times the sort location divided by the number of data points. Thus the highest value will be given a value of 100 and the lowest a value of 1.

Save Contour Image with Oonline

Checking this box will allow a displayed contour to be saved with a oneline diagram. If a contour is saved with a oneline diagram, the next time the oneline diagram is opened the contour will automatically be redrawn as well. The displayed contour is saved as a Bitmap image.

Continuously Update Contours

Normally contouring is only done on a snap shot of the power system state. However, you can also set PowerWorld to automatically update the contour every time the display is redrawn. In this way, an animation of the contour can be created. If you would like to create this animation, simply check the Continuously Update Contours checkbox. Note, however, that this will slow down the animation of the display, as the program must recalculate the contour at each step. If this slows down your display too much, try lowering the contour resolution to speed it up.

Note Regarding Values

Contours of most values create an image where the color around a data object is primarily related to the value of only that object. Some values however create "density-like" contours, where the color is related to the sum of the data object's values nearby. These include:

- Bus / Load MW
- Bus / Load Mvar
- Bus / Load MVA
- Bus / Cust Expr (Density)
- Area / Pos Spin Reserve
- Area / Neg Spin Reserve

Contour Type Options

Object

Simulator can contour several different values. To specify what Simulator should contour, first choose the type of display object; the options are Bus, Line, Interface, Area, Generator, Substation, Zone or Injection Group. Once the object is chosen, you must then choose the numerical value to be contoured by specifying it in the **Value** dropdown box.

If multi-section lines objects exist on the oneline diagram and the type of object to contour is Line, the contour for the multi-section line object will reflect the series of AC lines that it represents. If the type of object to contour is Bus, the locations of intermediate buses will be contoured.

Note: to contour a value for a type of object, representations of that type of object must be present on the oneline diagram. Choosing to contour an object type that is not represented on the diagram will result in no contour being drawn on the diagram.

Value

Select the quantity to contour from the Value dropdown box or click the **Find...** button to find the desired field (it is normally easier to use the Find button). Note: see [Contour Type](#) for more information.

Filter Object Values

The name of the advanced filter that is currently applied to the contour. Click Define to create a new [Advanced Filter](#) to use during contouring. Applying a filter means that *only* objects that meet the filter will affect the contour image. Objects which do not meet the filter will be ignored.

Influence Region

This track bar determines how far away each data point influences the contour image. A larger influence region results in each data point influencing more of the contour at the expense of longer screen refresh times.

Use Dynamic Influence Region

Dynamic influence distance determines how far out the contour should go when determining which buses influence the contour value for a screen point. The actual distance is the minimum of either 1) a common value for all screen points that depends upon user parameters [e.g., the dynamic region points value]), and now 2) the distance that includes the number of buses associated with the dynamic influence field. The primary reason for this option is speed, particularly when zoomed in on dense portions of the display. You should not see much impact on the contour itself.

Kind of Value

This option allows you to choose to contour based on the Actual Value or the Density Value. The Actual Value uses the weighted average of the data for computing the contour. The Actual Value method is most commonly used for contouring in Simulator. On occasion, the Density of Values method does not work as well. One example is contouring generator MW values. If you have four buses in close proximity, each with 100 MW of generation, and compare the contour with a single bus with 400 MW of generation, the contour based on the weighted average will look drastically different, despite the amount of generation being the same in each region. Using the Density Value option will correct the disparity, and the contour around these two different groups of generation would look basically the same.

Use Fade to Value

Checking this check box will allow to use the Fade to Value and Begin Fade Percentage options.

Fade to Value

The value to which a data point's value fades as it moves away from its location.

Begin Fade Percentage

While moving away from a data point, the data point's value decays towards the "Fade to" Value. The Begin Fade Percentage specifies when the contour starts to fade as a percentage of the largest distance for which this data point influences the contour.

Contour Resolution

This value determines the relative resolution of the contour. Increasing the contour resolution increases the level of detail represented on the map but will lengthen screen refresh times. Reducing the screen refresh time will yield less detail and shorter screen refresh times.

Low Resolution Display when Zooming or Panning**Do Low Resolution Speedup**

Check this option to enable drawing contours in low resolution while zooming or panning. Low resolution images are drawn much faster than high resolution images.

Low Resolution Normalized Percent

Set a percentage of resolution between 0.01 (1%) and 1 (100%). A smaller value translates to a lower resolution image.

Time Delay in Seconds before Redrawing at Full Resolution

Set the number of seconds Simulator will pause before returning the contour to full resolution following zooming and panning.

Minimum Resolution in Pixels for Low Resolution Displays

Specify the minimum resolution allowed (in pixels) when Simulator draws the low resolution image.

When Animating, Only Update Contour if Something Changed

As it describes, checking this option will suppress redrawing the contour during animation unless one or more of the contoured values changes.

Custom Color Map

Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the Custom Color Map Tab.

Reverse Color Map Colors

This check-box reverse the mapping of the color map, converting the colors corresponding to the high values into the colors for the low values, and vice-versa.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Make Discrete Color Map

Check this check-box to make a discrete color map, that is without having smooth transitions between colors.

Contour Type Values to Use

These check-boxes signify which values from the Contour Type Values Tab are used by the Color Map. There must be at least two contour type values checked.

Color Grid

The color grid on the right side of this page allows you to change the colors for each percentage breakpoint. In addition, you can add or delete breakpoints as well.

To change the color for a specific breakpoint, simply left-click on the color for the breakpoint you want to change. The Color dialog will open, and you can choose a new color for that breakpoint.

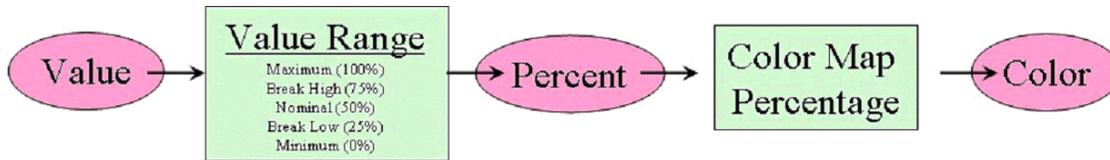
To add a breakpoint, right-click on a breakpoint position above or below where you would like the new breakpoint inserted. A popup menu will open, and you can select Add Above or Add Below, depending on where you wish the new breakpoint to be. Simulator will insert the breakpoint, and will automatically set the color and percentage at the midpoint between the two breakpoints above and below the inserted breakpoint. You can then click on the color to change it, or click on the percentage to type in a new value. The new percentage value should be between the values of the adjacent breakpoints.

To delete a breakpoint, right-click on the breakpoint you wish to delete, and choose Delete from the popup menu.

Color By...

This option allows the user to interpret the breakpoints values as percentage values or as direct values.

For all color maps that come with PowerWorld by default, the conversion of a value to a color is done in two steps. The value is first interpolated to a percentage by using up to 5 user-specified values (maximum, break high, nominal, break low and minimum) that correspond to specific percentages (100%, 75%, 50%, 25%, and 0%). This percentage is then interpolated through the color map and a color is determined.



Color maps which directly translate values into colors without the intermediate calculation of a percent may also be created. This is done by setting Color By to *Value*.

Save As...

To save the present color map as a new color map, click this button. Then specify a name for the new color map.

Save

To save changes that have been made to the present color map, click this button.

Rename

To rename the present color map, click this button.

Delete

To delete the present color map, click this button.

Store Color Maps in File

To store all custom color maps in a file for loading into another case, click this button. If you have saved any custom color maps with the current case, you will be prompted to choose a file name and location for saving the custom color maps. Color Maps are saved in the [Auxiliary File Format](#).

Load Color Maps from File

If you have created custom color maps in a different case and saved them to a file, you can click this button to load those color maps into your current case. Color Maps are loaded in the [Auxiliary File Format](#).

Functional Description of Contour Options

Functional Description of the Contour Options

The previous help topics discussed basic contour options in the order they are arranged on the dialog. The dialog is arranged so that the most important options are on the first tab and other on the Contour Type Options tab. This help topic discusses the options in a manner which better describes how the contour is actually created. A contour is calculated generally by a five-step process

1. Build a list of all possible data points (graphical locations)
2. Remove items that meet criteria from the list of data points
3. Assign a value to each data point
4. Calculate "virtual values" on a grid of points
5. Convert each "virtual values" into a color to create the contour image

Step 1: Build a list of data points (graphical locations)

The following options determine a list of potential data points (could also be called graphical locations) which will be used to calculate the contour image.

Object - Simulator can contour several different values. To specify what Simulator should contour, first choose the type of object. This corresponds to the type of display object that is drawn on the oneline diagram. For instance if you want to contour a substation value, then you must have substations drawn on your diagram. This selection narrows the set of quantities that can be contoured, which is specified in the Value dropdown box.

Data points Per Line - The contouring algorithm for lines is no different than for points, except that each line is represented by several points. For objects which are represented by graphical lines, this option will specify the number of data point which should be used to represent the line.

Step 2: Remove items that meet criteria from the list of data points

After a complete list of potential data points is made, there are then several options for filtering out things from this list which you do not want to effect the calculation of the contour image.

Filter - If you wish for the contour to be limited to only certain devices that meet specific criteria, click on this button to define an Advanced Filter for the contour

Ignore Above Max - Check this check-box to completely ignore values above the maximum percentage. This means that data which is larger than the Max % will not be used in calculating the contour image

Ignore Above Min - Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image

Ignore Zero Values - Check this check-box to completely ignore values that are zero.

Step 3: Assign a value to each data point

After Step 2, a list of potential data points has been created and a value must now be assigned to each data point. The following options specify how this is done.

Value - select the quantity to contour from the Value dropdown box or click the Find Value button to find the desired field.

Use absolute value - This check-box will modify the Value specified so that it uses the absolute value.

Interpretation – When interpretation is set to either Standard Deviations or Percentiles, then the value of the data point will be modified.

If **Standard Deviations** is chosen, then all the data point values will be used to determine a mean and standard deviation. The data point values will then be converted to represent the number of standard deviations they are from the mean. Thus a value equal to the mean will be changed to a 0, a value 1.5 standard deviations higher than the mean will be changed to 1.5, and so on. Similar,

If **Percentiles** is chosen, then all the data point values will be sorted from lowest to highest. The value will then be set equal to the 100 times the sort location divided by the number of data points. Thus the highest value will be given a value of 100 and the lowest a value of 1.

Step 4. Calculate "virtual values" on a grid of points

After Step 3, we now have a list of data points and a value assigned to each data point. We now must create a grid of points that represents "virtual values" throughout the entire graphical space. The values on this grid will be calculated based on the data point values

Contour Resolution - setting determine the size of the grid of points which will be superimposed on the present oneline diagram. The higher the resolution the more number of grid points will be used to represent the contour (and thus make the calculation of the contour image slower)

Influence Region - each data point value will effect only the grid locations that are "near" it. The distance that is considered "near" is determined by the Influence Region. Setting the influence

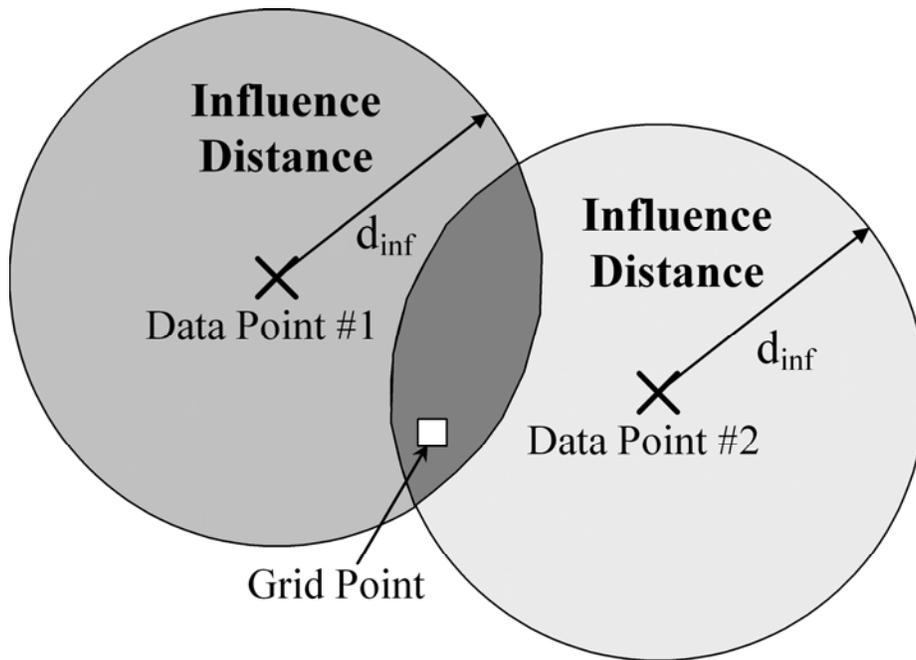
region higher will result in each data point effecting a larger portion of the contour image (and thus making the calculation of the contour image slower)

Kind of Value - The calculation of the virtual value at a particular grid point is done by first building a list of data points that are within the "influence distance" of the grid point. The Kind of Value setting determines what process is used to calculate the virtual value from this list of values.

Actual Value (Weighted Average) - The virtual value is calculated as the weighted average value, weighted by the distance from the grid point. This means that virtual value half-way between two data points will be the average of the two data point values.

Density of Values (Weighted Sum) - The virtual value is calculated as the sum of value that within the influence region. This means that virtual value half-way between two data points will be the sum of the two data point values.

Actual Value (Only Closest) - The virtual value is assigned as the value of the closest data point.



Fade to Value and Begin Fade Percentage - In the middle of a contour image the colors look consistent because the virtual values are calculated using a good number of data points which surround it. Sometimes at the edge of a contour image however, the colors can become skewed because there are very few data points influencing it. Using a fade to value can help this situation, but will skew the entire contour image in general instead. Without using the fade to value, the raw data point values will be used when calculating virtual value.

When using the fade to value, while moving away from a data point, the data point's value decays towards the "Fade to" Value. The Begin Fade Percentage specifies when the contour starts to fade as a percentage of the largest distance for which this data point influences the contour.

Step 5. Convert each "virtual values" into a color to create the contour image

After Step 4, a grid of virtual values has been calculated. At this point, must specify how these numbers map to colors. This done through the user of a Color Map.

Color Map - Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image. A user may also define additional color maps by going to the Custom Color Map Tab.

Brightness - this value is used to brighten or darken the colors specified in the color map.

Values - These values along with the color map define how to convert your values into a color for the contour. The values are:

Maximum - The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Break High - This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

Nominal - This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

Break Low - This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

Minimum - The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Interpretation - Most frequently this option will be set to **Fixed Values** meaning values for maximum, break high, etc... are entered directly in the units of the value being contoured. If Interpretation is set to **Standard Deviations** or **Percentiles** then as mentioned earlier the values mean something different. The options **Dynamic Values** will process this list of data point values and automatically set the values as follows: Maximum = Maximum data point value; Minimum = Minimum data point value; Nominal = Average data point value; Break High = $(\text{Max} + \text{Average})/2$; and Break Low = $(\text{Min} + \text{Average})/2$.

No Data Color - It is likely that some the grid points will not be within the influence of any of the data points. The color of these points is determined by no data color setting.

Power Transfer Distribution Factors

The Power Transfer Distribution Factor (PTDF) display is used to calculate the incremental distribution factors associated with power transfers between two regions. These regions can be defined by areas, zones, super areas, single buses, injection groups or the system slack. These values provide a linearized approximation of how the flow on the transmission lines and interfaces change in response to transaction between the Seller and the Buyer. These values can then be visualized on the onelines using animated flows (see below for details). When considering areas, zones, or super areas as part of the transaction, the transaction for which the PTDFs are calculated is modeled by scaling the output of all generators on [AGC](#) in the source and sink areas in proportion to their relative [participation factors](#). Generators in the source area increase their output, while generators in the sink area decrease their output. When including injection groups as part of the transaction, the AGC status of any participating generators does not matter. All generators in the injection group with non-zero participation factors will be included in the transaction. When loads are included in injection groups, loads in the Seller would be decreased in order to effect a transaction and loads in the Buyer would be increased in order to effect a transaction. Of course, the PTDF calculation is a linear calculation and no changes are actually made to the output of generators or loads to study the impact of the transaction.

An important aspect to consider in calculating the PTDF is how the losses associated with the transfer are allocated. Simulator assumes that the Seller increases its injection by 100% of the transfer amount (generators increase and loads decrease), while the Buyer decreases its injection (generators decrease and loads increase) by 100% of the transaction **minus any change in system losses**. In other words, the Buyer accounts for the entire change in the system losses. Of course it is possible that a transfer may result in decreased system losses; for that case, the Buyer's change in injection will be greater than 100% of the transfer.

To Calculate the Power Transfer Distribution Factors:

- Perform an initial Power Flow Solution.
- In Run Mode, click on the Tools ribbon tab and select **Power Transfer Distribution Factors (PTDFs)** from the **Sensitivities** menu on the **Run Mode** ribbon group to open the [Power Transfer Distribution Factors Dialog](#).
- Supply the requested information on the [Power Transfer Distribution Factors Dialog](#) and click the **Calculate PTDFs** button. The distribution factors are calculated and displayed for the element set of your choice in the table at the bottom of the dialog.

The animated flows that appear on the oneline diagram may represent either actual flows or PTDF values. To specify that the display should show distribution factors, click the button labeled **Visualize PTDFs**. Once this button is clicked, the flow arrows on all open onelines will represent distribution factors, any [transmission line pie charts](#) will show PTDF values, and the caption of the button will change to **Visualize Actual Power Flows**. Click the button again to visualize actual power flows instead of distribution factors.

Note that when calculating PTDF values for interfaces that include contingent elements, the PTDF values reported are actually what are referred to as an Outage Transfer Distribution Factor (OTDF). See [Line Outage Distribution Factors \(LODFs\)](#) for more information.

Power Transfer Distribution Factors Dialog

The PTDF Dialog enables you to control and to view the results of [power transfer distribution factor calculations](#). To access this dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Power Transfer Distribution Factors** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The dialog has the following options:

Linear Calculation Method

PTDFs may be calculated using either the full power flow Jacobian or only a portion of it. If you select the **Linearized AC Approximation** option, the sensitivity of the monitored element's flow will be calculated as a function of both its real and reactive power components to the voltage magnitude and angle of its terminal buses. When using the AC method, losses are included in the calculation. Simulator assumes that the change in losses is taken care of by the Buyer.

If you instead select the **Lossless DC Approximation** option, branch flow sensitivity is calculated by estimating the real power that flows through the monitored element only from the difference in angles measured across its terminals. This method assumes that there are no losses.

The **Lossless DC with Phase Shifters Approximation** option is similar to the **Lossless DC**, except additional constraints are placed on the calculations that assume that the change in flow across active phase-shifting transformers is zero.

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

Directions

This option allows you to choose to define a single direction PTDF using the **Seller** and **Buyer Type** related fields, or to define multiple transfer directions between many different entities. If you choose to use multiple directions, the **Seller** and **Buyer Type** fields are replaced by the [Direction Records display](#) for viewing and defining directions.

For Single Direction: Seller Type, Buyer Type

Distribution factors can be calculated for power transfers between combinations of areas, zones, super areas, participation groups, or to a slack bus. Use the *seller type* and *buyer type* options to indicate the type of the selling and purchasing entities. These fields are only present for single direction PTDFs. When choosing Area, Zone, or Super Area, this really means all generators in the Area, Zone or Super Area weighted by the generator participation factor.

For Single Direction: Seller, Buyer

These drop-down boxes allow you to select the selling and buying entities. Their contents are filled when you select the seller and buyer types. These fields are only present for single direction PTDFs.

For Single Direction: Reverse Buyer/Seller

Click this button to re-calculate PTDFs for the direction that is the reverse of the direction currently shown. For example, if you have just calculated PTDFs for a transaction from area A to area B, press this button to calculate and display PTDFs for a transaction from area B to area A. This option is only present for single direction PTDFs.

Calculate PTDFs

Click on this button to update the PTDF values. The results table will reveal the latest calculations.

Automatically Recalculate

If checked, the PTDFs are automatically recalculated every time the power flow is solved.

Calculate MW-Distance

Click this button to open the **MW * Distance Calculations** form. This form allows you to calculate MW * Distance values for the transaction for which you calculated PTDFs. See [MW-Distance Calculations](#) for more information.

Increase in Losses

This is a read-only field that indicates the change in system losses caused by the transfer from the selling area to the buying area. The change is expressed as a percentage of the transfer amount. This will only be non-zero when using the **Linearized AC** calculation method.

Use Area/Zone Filters

If this box is checked, then the results table at the bottom of the dialog will include only records associated with devices located in areas or zones included in the [area/zone/owner filter](#) set.

Only Show Above %

Restricts the result set to show only those PTDFs that exceed a specified value.

Visualize Actual Power Flows, Visualize PTDFs

Select to toggle the onelines between showing the actual power flows and the PTDF flows. Selecting this button changes the Flow Visualization field for all the visible onelines. You can also change this field manually using the [Online Display Options](#) Dialog. When choosing to visualize PTDFs, the animated flow arrows will represent PTDF values, and any [transmission line pie charts](#) will also be changed to show PTDFs.

Tables of Results

The tables of results occupy the bottom of the PTDF Dialog. They are a set of [case-information displays](#) and thus share many characteristics common to all other case information displays. The tables will show results for lines/transformers, interfaces, areas, zones, generators, and phase shifters. The tables feature a [local menu](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated element. You can also sort the area records by clicking on the heading of the field by which you want to sort.

Lines/Transformers

Shows the transaction distribution factors for the lines and transformers. The following fields are shown:

From Number, From Name, To Number, To Name, Circuit

Identifiers for the transmission line or transformer.

% PTDF From

Distribution factor associated with the MW flow at the "from bus" end of the line or transformer, specified as a percentage of the transaction amount.

% PTDF To

Distribution factor associated with the MW flow at the "to bus" end of the line or transformer, specified as a percentage of the transaction amount.

% Losses

Shows the percentage of the PTDF assigned as losses.

Nom KV (Max) and Nom KV (Min)

Displays the maximum and minimum nominal voltages for the line. This is useful for identifying transformers and which end of the PTDF relates to which nominal voltage.

Interfaces

Shows the transaction distribution factors for the [interface records](#). The following fields are shown:

Name

Alphanumeric identifier for the interface.

Number

Numeric identifier for the interface.

% PTDF

Distribution factor associated with the MW flow through the interface, specified as a percentage of the transaction amount. A positive value indicates the transaction would result in an increase in the flow through the interface.

Interface MW Flow

Amount of real power flowing on the interface.

Has Contingency

Signifies if an element of the interface is violating a limit.

Areas and Zones

Shows the impact the transaction has on the losses for the area or zone. The following fields are shown:

Area/Zone Number and Name

Number and name identifiers for the area or the zone

Losses %

Change in the losses in the area or zone, specified as a percentage of the transaction amount. A **positive number** indicates that the transaction would result in **increased** losses in the area or zone, while a **negative number** indicates that the transaction would result in **decreased** losses.

Gen Chg %

Total change in all of the generators in area or zone, specified as a percentage of the transaction amount. For areas, this field should show 100% in the selling area, and 100% minus the change in system losses in the buying area.

Generators

Shows the marginal participation of each generator in the transaction. The following fields are shown:

Number of Bus, Name of Bus, ID

Generator's terminal bus number and alphanumeric identifier, and the id for the generator.

Area Num of Gen, Area Name of Gen

Name and number of the generator's area.

Gen Chg %

Assumed participation of the generator in the transaction, specified as a percentage of the transaction amount. This value is directly proportional to the participation factor for the generator, provided the generator is available for AGC and is free to move in the specified direction (i.e., is not at a MW limit). The generator's participation factor and AGC status are modified on the [Generator Dialog](#), which can be displayed by right-clicking anywhere in the record's row in the table and selecting the *Show Dialog* option.

Phase Shifters

Shows the transaction distribution factors for the phase shifters. This applies when using the Lossless DC with Phase Shifters calculation method. The following fields are shown:

From Number, From Name, To Number, To Name, Circuit

Identifiers for the phase shifter.

Status

Indicates whether or not the phase shifter is in-service.

Phase (Deg)

The current phase angle of the transformer.

XF Auto

Indicates if the phase shifter is currently enabled for automatic control.

Deg per MW

This field indicates the amount of angle change, in degrees, that would be required to keep the flow across the transformer constant for a one MW transfer between the seller and the buyer.

Tap Min, Tap Max

The minimum and maximum tap positions for the phase shifter.

Line Outage Distribution Factors (LODFs)

Line Outage Distribution Factors (LODFs) are a sensitivity measure of how a change in a line's status affects the flows on other lines in the system. On an energized line, the LODF calculation determines the percentage of the present line flow that will be show up on other transmission lines after the outage of the line. For example, consider an energized line, called LineX, whose present MW flow is 100 MW. If the LODFs are found to be

LODFs for LineX outage

LineX -100%

LineY + 10%

LineZ - 30%

This means that after the outage of LineX, the flow on LineX will decrease by 100 MW (of course), LineY will increase by 10 MW, and LineZ will decrease by 30 MW. The "flow on Line X" here means the flow at the *from* bus going toward the *to* bus.[Line Outage Distribution Factors \(LODFs\)](#)

Similarly, sensitivities can be calculated for the insertion of a presently open line. In this case, the LODF determines the percentage of the post-insertion line flow that will come from the other transmission line after the insertion. The "LODF" might better be named a Line Closure Distribution Factor (LCDF) in this case.

To calculate the LODFs:

- Perform an initial [Power Flow Solution](#).
- In [Run Mode](#), select **Sensitivities > Line Outage Distribution Factors (LODFs)** from the **Rune Mode** ribbon group on the [Tools](#) ribbon tab to open the [Line Outage Distribution Factors Dialog](#).
- Supply the requested information on the [Line Outage Distribution Factors Dialog](#) and click the **Calculate LODFs** button.

What else are LODFs used for?

LODFs are used extensively when modeling the linear impact of contingencies in Simulator. This is true for the calculation of [PTDFs](#) for interfaces which contain a contingent element, as well as when performing Linear [ATC analysis](#) that includes branch contingencies.

When calculating "PTDF" values for interfaces that include [contingent elements](#), the PTDF values reported are actually what are referred to as an Outage Transfer Distribution Factor (OTDF). An OTDF is similar to PTDF, except an OTDF provides a linearized approximation of the *post-outage* change in flow on a transmission line in response to a transaction between the Seller and the Buyer. The OTDF value is a function of PTDF values and LODF values. For a single line outage, the OTDF value for line x during the outage of line y is

$$\text{OTDF}_x = \text{PTDF}_x + \text{LODF}_{x,y} * \text{PTDF}_y$$

where PTDF_x and PTDF_y are the PTDFs for line x and y respectively, and $\text{LODF}_{x,y}$ is the LODF for line x during the outage of line y. More complex equations are involved when studying contingencies that include multiple line outages, but the basic idea is the same.

When performing Linear [ATC analysis](#) along with calculating OTDFs, Simulator determines the linearized approximation of the post-outage flow on the line. This is similarly determined as

$$\text{OutageFlow}_X = \text{PreOutageFlow}_X + \text{LODF}_{x,y} * \text{PreOutageFlow}_Y$$

where PreOutageFlowX and PreOutageFlowY are the pre-outage flow on lines x and y.

Line Outage Distribution Factors Dialog

The LODF Dialog enables you to control and to view the results of Line Outage Distribution Factor calculations. You access this dialog by selecting **Sensitivities > Line Outage Distribution Factors (LODFs)** from the **Run Mode** ribbon group on the [Tools](#) ribbon tab in [Run Mode](#) only.

The dialog has the following options:

Output Option

Choose Single LODF to calculate only the sensitivities related to the status change of a single transmission line. Choose LODF matrix to calculate a matrix of sensitivities.

For Single LODF: Specify Near Bus, Far Bus

Specify the line whose status modification you would like to determine sensitivities to. The direction of the near/far chosen is important. Switching the near and far bus around will change the sign of all the LODF sensitivities.

For Single LODF: Action

Check Outage Sensitivities to determine sensitivities for the outage of a line.

Check Closure Sensitivities to determine sensitivities for the closure of a presently outaged line. The LODF might better be named a Line Closure Distribution Factor (LCDF) in this case.

For LODF Matrix: Specify Lines to Process (Outage or Closure) and Lines to Monitor

Choose which lines to process and which lines to monitor by choosing either all AC lines, only those that meet the area/zone/owner filters, only lines with Selected field set to YES, or only those lines that meet an [Advanced Filter](#). After choosing **Calculate LODFs**, each row of the results represents the line being outaged/closed. Each column of the results represent the line begin monitored and the corresponding LODF/LCDF for that line.

Linear Calculation Method

Linearized AC

This calculation method is not available for LODF sensitivities.

Lossless DC

Uses the DC power flow approximation.

Lossless DC with Phase Shifters

Check this to include the impact of phase shifter controllers with the Lossless DC calculation. By checking this, it is assumed that operating phase shifters will maintain their control requirements after the line outage.

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

Calculate LODFs

Click this to calculate the LODFs and update the display.

Advanced LODF Calculation

Opens the [Advanced LODF Calculation](#) dialog for setting additional options for the calculation.

LODFs Tab

After calculating a LODF Matrix, each row of the results represents the line being outaged/closed. Each column of the results represent the line begin monitored and the corresponding LODF/LCDF for that line.

When calculating LODFs for a single branch, This tab contains a table showing a list of the lines in the case. Since this table us another variety of the [Case Information Displays](#), you may interact with it in a familiar manner. Click on any of the field headings to sort by that field. Right-click on the display to call up the display's [local menu](#). From the [local menu](#), you can print the violations, copy the violation records to the Windows clipboard for use with another application, modify the format and content of the listing, view the information dialog of the respective element, and view the [Quick Power Flow List](#) or [Bus View Display](#).

The default fields for the tab are as follows

From Bus Number and Name

"From" bus number and name. Right- clicking on either of these fields allows you to see all the flows measured at the "from" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

To Bus Number and Name

"To" bus number and name. Right-clicking on either of these fields allows you to see all the flows into the "to" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses.

%LODF

The LODF value for the line.

From MW, To MW

The present MW flows on the line at the "from" bus and the "to" bus.

From CTG MW, To CTG MW

The projected MW flows after the change in line status on the line at the "from" bus and the "to" bus.

Note: LODFs are always calculated using a DC power flow technique.

Advanced LODF Calculation Dialog

The Advanced LODF Calculation Dialog can be opened by clicking on the Advanced LODF Calculation button found on the [Line Outage Distribution Factors \(LODFs\) dialog](#). The advanced calculation dialog allows you to calculate the [LODFs](#) for several different contingent lines in one batch process. First, you need to use the [Contingency Analysis Tool](#) to define which contingent lines to use (all contingencies that contain a single branch outage will be used.) Also, only lines that are being monitored will have their LODFs calculated. See [Limit Monitoring Settings](#) to change which lines are monitored.

The output from the advanced LODF calculation process is saved to a file for import into another program for analysis. This dialog defines how to save the Advanced LODF results to a file for importing in another program.

Format to Save in

You can choose to save the advanced LODF results as "Monitored Branch, Contingency" pairs for PROMOD, or as a matrix in a comma-delimited text file. The comma-delimited text file is useful for loading the results into a spreadsheet program.

Only save pairs with an LODF whose absolute value is greater than

This field allows you to filter out elements whose LODF is below a certain value.

Only Include Monitored Branches whose MW flow increases

This field allows you to filter out monitored elements whose MW flow did not increase in the LODF calculation.

Maximum Columns Per Text File

This field is important when you are saving a matrix in a comma-delimited text file. Most spreadsheet programs have limits on the number of columns they can display. The default is 256, which happens to be the maximum number of columns allowed in Excel.

LODF Number Format

You can choose to have the LODF values stored in either scientific notation, or as a specified length decimal number.

File Name

You must enter a file name and location or **Browse** for a file for saving the LODF data.

Directions Display

The Directions Display appears in the upper-right corner of the PTDF window when you select Multiple Directions for the PTDF Type. The Directions Display is a [case information display](#) that allows directions to be defined for performing multiple direction [Power Transfer Distribution Factors](#). The Directions Display allows you to insert, delete, and modify directions using options available from the display's [local menu](#). When directions are modified or inserted individually, the Directions Dialog will be displayed for entering the information. In addition to individually defining directions, they can also be automatically inserted using the Auto Insert Directions dialog.

Each record in the display shows the following default information:

Number, Name

A unique number and name given to the defined direction.

Source Type, Source Name

The name and type of the direction source.

Sink Type, Sink Name

The name and type of the direction sink.

Include

Determines if the direction is to be analyzed when calculating the multiple direction PTDFs.

Processed

Displays if the direction has already been processed for a multiple direction PTDF analysis.

Directions Dialog

Directions are objects that are defined and used when computing a [Power Transfer Distribution Factor](#) using multiple directions.

The **Directions Dialog** can be used to insert a new direction or to modify the information for an existing direction. This dialog can be called by choosing **Insert** or **Show Dialog** from the [Directions Display](#) local menu.

The options that can be set from this dialog include:

Name of Direction

The name for the direction. If you are entering a new direction, you can enter a new direction name. If you are modifying an existing direction, the name of the currently viewed direction will be displayed. A drop down list shows a list of currently defined direction names, and choosing one from the list will display that direction's information in the dialog. The up and down arrows next to the field also allow you to scroll through the list of defined directions.

Direction Number

If you are entering a new direction, you can put in a new direction number. If you are viewing or modifying an existing direction, the direction's number will be displayed.

Source

Select the type of the transfer direction source. Once the type has been selected, the [advanced find](#) list will be enabled, allowing you to search for the specific object or group of objects to set as the direction source.

Sink

Select the type of the transfer direction sink. Once the type has been selected, the [advanced find](#) list will be enabled, allowing you to search for the specific object or group of objects to set as the direction sink.

Include in list of monitored directions

Determines if the direction is to be analyzed when calculating the multiple direction PTDF's.

Auto Insert Directions

Multiple directions can be automatically inserted for PTDF studies using the **Auto Insert Directions** dialog. This option can be selected from the local menu of the [Directions Display](#) by right-clicking and choosing **Direction Records > Auto Insert Directions**.

The layout of the dialog is as follows:

Type of Direction

There are six types of directions that can be automatically defined. **Area to Slack**, **Zone to Slack** and **Injection Group to Slack** will define directions from areas, zones or injection groups to the slack bus of the system. **Area to Area**, **Zone to Zone** and **Inj. Group to Inj. Group** will define directions from areas to other areas, zones to other zones, or injection groups to other injection groups.

Delete Existing Directions

When checked, any previously defined directions will be deleted before the new directions are automatically inserted. If not checked, then automatically inserted directions will be added to the list of previously defined directions. By not deleting existing directions before automatically inserting new directions, it is possible to have more than one direction defined with the same source and sink.

Only Insert for Areas or Zones with Display Filters Set

If checked, then only Areas and Zones with their [Area/Zone/Owner Filters](#) set to **Yes** will be used when automatically inserting directions.

Starting Number, Increment By

The **Starting Number** will be the first number used when automatically numbering the automatically inserted directions. Each subsequent direction added will be numbered according to the **Starting Number** and the **Increment By** value.

Insert Directions

Insert Directions will perform the automatic insertion routine for the directions, according to the defined options.

Flow and Voltage Sensitivities

To access the Flows and Voltages Sensitivities dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Flows and Voltages Sensitivities** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

Single Meter, Multiple Transfers

The Single Meter, Multiple Transfers Tab of the Flows and Voltages Sensitivities Dialog shows the effect an additional injection of real or reactive power at a bus has on real, reactive, or complex power flow on a particular line or interface, or on the voltage of a selected bus. The grid that occupies the bottom of the dialog lists each bus in the system, subject to the [Area/Zone/Owner Filter](#) settings. This grid is a [case information display](#) and thus shares properties and controls common to all other case information displays. The *P Sensitivity* field indicates the effect a 1 MW increase in real power at the bus has on the flow (either MW, Mvar, or MVA flow, as dictated by the **Flow Type** setting) or voltage on the device identified by the **Device Identifier**. Likewise, the *Q Sensitivity* field indicates the effect a 1 Mvar increase in reactive power at the bus has on the flow (either MW, Mvar, or MVA flow, as dictated by the **Flow Type** setting) or voltage (in per unit) on the device identified by the device identifier. All injections assume that the power is absorbed at the island slack bus.

Use the **Device Type** control to indicate whether the sensitivities are to be calculated for a line/transformer, an interface, or a bus. Use the **Flow Type** control to specify the type of power flow for which the sensitivities will be calculated. Finally, select the line/transformer, interface or bus.

Whenever you make a change to any of these settings, click **Calculate Sensitivities** to update the grid with the new sensitivities.

The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Injection dValue/dP* gives the sensitivity of per unit voltage magnitude to an injection of real power at the bus chosen in the Select Device section.
- *Sensitivity: Injection dValue/dQ* gives the sensitivity of per unit voltage magnitude to an injection of reactive power at the bus chosen in the Select Device section.
- *Sensitivity: Injection dValue/dVsetpoint (for PV bus)* gives the sensitivity of per unit voltage magnitude to an injection of reactive power at the bus chosen in the Select Device section.
- *Sensitivity: Injection dAngle/dP (radians/MW)* gives the sensitivity of voltage angle magnitude to an injection of real power at the bus chosen in the Select Device section.
- *Sensitivity: Injection dAngle/dQ (radians/MVar)* gives the sensitivity of voltage angle to an injection of reactive power at the bus chosen in the Select Device section.

The option labeled **Set Out-Of-Service** allows you to approximate the sensitivities at out-of-service buses with the sensitivity of the closest bus. Otherwise the out-of-service buses will display sensitivities of 0 when the sensitivities are calculated.

Single Transfer, Multiple Meters

The Single Meter, Multiple Transfers Tab of the Flows and Voltages Sensitivities Dialog show the effect of a single transfer of real or reactive power on all the voltages in the system. To specify the transfer choose the **Seller** and **Buyer** in the same manner done on the [Power Transfer Distribution Factors](#) dialog. The specify whether the calculation of Real Power (P), Reactive Power (Q), or both are of interest.

Click **Calculate Sensitivities** to perform the calculations.

After clicking **Calculate Sensitivities**, each row of the results gives the sensitivity of voltage at the respective bus to the power transfer defined by the seller and buyer. The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Transfer dV/dP* gives the sensitivity of voltage to a real power transfer
- *Sensitivity: Transfer dV/dQ* gives the sensitivity of voltage to a reactive power transfer
- *Sensitivity: Transfer $dAngle/dP$* gives the sensitivity of voltage to a real power transfer
- *Sensitivity: Transfer $dAngle/dQ$* gives the sensitivity of voltage to a reactive power transfer

The check box option **Turn off generator AVR control for generators participating in transfer** is also available. Selecting this option will turn off AVR control for only those generators participating in the transfer with a non-zero participation factor, resolve the power flow, calculate the voltage sensitivities, turn the AVR control back on for those generators for which it was changed, and then resolve the power flow. Stated another way, this option assumes that all generators maintain a constant reactive power output and do not maintain their voltage setpoint.

Self Sensitivity

The Self Sensitivity Tab of the Flows and Voltages Sensitivities Dialog shows the affect an of an injection of real or reactive power at a bus on its own voltage (assuming everything is absorbed at the island slack bus). It processes through each bus calculating its own self-sensitivity only.

Each row of the results gives the sensitivity of voltage at the respective bus to a power injection at the same bus. dV/dP gives the sensitivity of voltage to real power injection and dV/dQ gives the sensitivity of voltage to reactive power injection. To speed up the calculation, sensitivities are calculated only for the buses displayed in the Bus Sensitivities grid. The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Self dV/dP (per unit/MW)* gives the sensitivity of voltage magnitude to an injection of real power.
- *Sensitivity: Self dV/dQ (per unit/MVar)* gives the sensitivity of voltage magnitude to an injection of reactive power
- *Sensitivity: Self $dAngle/dP$ (radians/MW)* gives the sensitivity of voltage angle to an injection of real power.
- *Sensitivity: Self $dAngle/dQ$ (radians/MVar)* gives the sensitivity of voltage angle to an injection of reactive power.

Click **Close** to close the Flows and Voltages Dialog.

Loss Sensitivities

To access the Loss Sensitivities dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Loss Sensitivities** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The **Bus Marginal Loss Sensitivities Dialog** is used to calculate and display the sensitivity of a real power loss function, P_{Losses} , to bus real and reactive power injections. Stated mathematically, the display calculates $d P_{Losses}/d P_i$ and $d P_{Losses}/d Q_i$, where P_i and Q_i are the real and reactive power injections at bus i , respectively.

Stated less formally, the display indicates how losses would change if one more MW or Mvar of power were injected at bus i . Simulator can calculate the losses for a bus relative to losses in the bus' island or area, to losses in a select group of areas, or, if the bus belongs to a super area, to losses in the bus' super area. How Simulator computes the losses is governed by the value of the **Loss Function Type** option.

The Loss Function Type may assume one of the following six values:

Do Not Calculate Bus Loss Sensitivities

No Losses are calculated because a loss function is not specified.

Each Electrical Island

Losses are calculated with respect to the losses in bus' island. If the power system consists of only one island, losses are computed with respect to the total system losses.

Each Area

Losses are calculated with respect to the total losses for the area containing bus i . This is probably the most common loss function because usually one is concerned with minimizing losses for a particular area rather than for the entire case.

Each Area or Super Area

Losses are calculated with respect to the total losses for the area containing bus i if bus i does not belong to a super area, and with respect to the total losses for the super area containing bus i if bus i does belong to a super area. When a case contains Super Areas this is more commonly done. When Simulator calculates loss sensitivities internally for use in the Economic Dispatch or the Optimal Power Flow calculation, this is the option which is used.

Areas Selected on Loss Sensitivity Form

Losses are calculated with respect to the total losses for a group of areas, specified in the Selected Areas table.

User-Specified

If you select User-Specified as the loss function type, the values last calculated using a different loss function type will become fixed. Thus you can force the loss sensitivities to remain constant when used in other features and tools of Simulator.

In steady-state power system operation, total generation must always equal total load plus losses. Therefore, the real power injection at a single bus cannot be changed arbitrarily; it must be met by a corresponding change somewhere else in the system so that the total power remains balanced. In other words, the change in power injection must somehow be absorbed. How the injection is absorbed depends on the Loss Function Type. If the Loss Function Type is **Each Island**, the injection is absorbed by the island slack. For the **Each Area** and **Selected Areas** loss functions, the injection is absorbed at the area tie-lines.

The loss sensitivities are calculated by modeling an injection of power at a bus and then assuming that this injection is absorbed by the island slack bus. The sensitivity then shows how much the losses (for the region of interest) increase when you transfer 1 MW at the injection bus to the island slack. The "region of interest" is what was chosen as Island, Each Area, or Selected Areas.

Therefore, the "absolute numbers" given by the loss sensitivity dialogs are not directly meaningful because we are always assuming that the absorbing point is the island slack bus. What is meaningful is the "difference" between sensitivity numbers.

Example:

Assume the sensitivities are calculated to be

Bus A Loss MW Sensitivity = -0.04 = A_{sens}

Bus B Loss MW Sensitivity = -0.02 = B_{sens}

Bus C Loss MW Sensitivity = +0.03 = C_{sens}

Using these we can then look at the sensitivity of generic transfers between these buses by using "superposition".

Consider the following change in injections modeling a transfer of power from Bus A to Buses B and C.

Bus A injection = +10 MW = AMW

Bus B injection = - 6 MW = BMW

Bus C injection = - 4 MW = CMW

An estimate of the change in losses can then be calculated as

$$\begin{aligned} \text{Loss Change} &= (AMW)(A_{sens}) + (BMW)(B_{sens}) + (CMW)(C_{sens}) \\ &= (+10)(-0.04) + (- 6)(-0.02) + (- 4)(+0.03) \\ &= -0.4 \text{ MW} \end{aligned}$$

The Bus Marginal Loss Sensitivities Dialog houses the following controls:

Selected Areas Table

This table is used only when the **Loss Function Type** is set to *Selected Areas*; otherwise, it is ignored. Left-click on the *Include* field to include or exclude areas from the loss function.

Calculate Marginal Loss Sensitivities Button

Once the loss function type has been specified, click this button to calculate the bus marginal loss sensitivities and update the Bus Marginal Loss Sensitivities table.

Bus Marginal Loss Sensitivities Table

This table shows the bus marginal loss sensitivities for all buses with valid [Area/Zone/Owner filters](#). The Bus Marginal Loss Sensitivities Table is a type of Case Information Display and thus exhibits features and behavior similar to all other case information displays. It has a local menu from which you can choose to find out more about a particular bus. You can sort records by any of the listed fields by clicking on the column headings. The table contains the following fields:

Number, Name Number and name of the bus.

Area Number, Area Name Number and name of the bus' area.

Loss MW Sens. Sensitivity of the loss function to an increase in the real power injection (generated power assumed positive) at the bus.

Penalty Factors Computed penalty factor of each bus.

MVR Sens. Sensitivity of the loss function to an increase in the reactive power injection (generated power assumed positive) at the bus.

Just Generators Marginal Loss Sensitivities Table

This table shows the bus marginal loss sensitivities for only the generator terminal buses with valid [Area/Zone/Owner filters](#). This table is otherwise identical to the displayed values and table operation as the Bus Marginal Loss Sensitivities Table described above.

Transmission Loading Relief Sensitivities

Transmission Loading Relief (TLR) Sensitivities are similar to the [Power Transfer Distribution Factors](#). Both TLR Sensitivities and PTDFs measure the sensitivity of the flow on a device to a transaction. To calculate PTDFs, you specify a source group and a sink group, and Simulator determines the percentage of a single transfer between the source and sink that flows on each of several monitored elements. For TLR sensitivities, you specify a single device, such as a transmission line, to monitor, and a group that serves either as source or as sink. Simulator then determines the sensitivity of the flow on the single monitored element to many different transactions involving the group you specified as the source or sink. To summarize, PTDFs express the sensitivity of many monitored elements to a single transaction, whereas TLR sensitivities gauge the sensitivity of a single monitored element to many different power transfers.

"TLR" stands for "Transmission Loading Relief." TLR is an industry-wide tool for managing transmission utilization to prevent overload situations that put the system at risk. For example, suppose a particular line is loaded beyond its thermal limit. Its owner will request that a TLR program be initiated, which dictates that all transactions for which 5% or more of the exchanged power flows on the overloaded element be curtailed. The TLR sensitivity tool in Simulator is useful for pinpointing those transactions that would be curtailed. Suppose we use the TLR tool to determine where area A can purchase power from while the TLR for the overloaded element is in place. We specify area A as the buyer area, identify the overloaded line, and tell Simulator to perform the calculation. Simulator will then list the sensitivity of the flow on the overloaded line to power exchanges between all other generators, areas, and buses to area A. Any transaction for which the sensitivity exceeds 5% would be curtailed; anything below 5% would be allowed to continue.

To calculate TLR Sensitivities, select **Tools > Sensitivities > TLR Sensitivities** from the main menu. This will open the [TLR Sensitivities dialog](#), which will allow you to set TLR Sensitivities options and then calculate the sensitivities.

TLR Sensitivities Dialog

The TLR Sensitivities dialog allows you to calculate [Transmission Loading Relief Sensitivities](#) for the load flow case at its solved load flow point. To access this dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > TLR Sensitivities / Generation Shift Factors** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The following describes the sections of the dialog:

Device Type

Select whether you want to calculate the sensitivities for a transmission line/transformer, for an interface, or for multiple elements. To calculate sensitivities for an interface, you must have the [interface defined](#) in the case.

When you choose an individual line/transformer or interface, you need to specify the device by selecting it from the list of devices. In this case you specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name or number. When you select the Multiple Elements option, the device selection area of the dialog changes to allow you to select a [TLR Multiple Device Type](#). The TLR results display changes accordingly in order to accommodate TLR sensitivities for multiple elements.

Line/XFMR or Interface Only: Choose device

Specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name, number, and monitored flow direction.

For multiple elements, specify the multiple device type. Choices are: selected lines/transformers, selected interfaces, overloaded lines/transformers in the base case, overloaded interfaces in the base, overloaded lines/transformers during the set of contingencies, and overloaded interfaces during the set of contingencies.

Multiple Elements Only: Select Lines/XFMRs

Opens the case information display for lines/transformers. To include a line/transformer in the TLR study, set the **Selected?** field to Yes.

Multiple Elements Only: Select Interfaces

Opens the case information display for interfaces. To include an interface in the TLR study, set the **Selected?** field to Yes.

Transactor Type

Specify if the sensitivities will be calculated for the transactor being the buyer or the seller.

Transactor Object

Specify what the transactor will be. The choices are [Area](#), [Zone](#), [Super Area](#), [Slack](#), [Injection Group](#), and [Bus](#). When choosing Area, Zone, or Super Area, this really means all generators in the Area, Zone or Super Area weighted by the generator participation factor.

TLR Sensitivities

Specify if the next set of calculated TLR sensitivities should replace the currently calculated values, or be appended to the current values. The normal option is to Clear before Calculate. When choosing to Append, then as new calculations are performed, the results will only replace the existing sensitivity value store if the new value is larger than the old value. (Note: in this situation, negative values are considered smaller than positive values).

PTDF Calculation Method

Choose the solution method to use for calculating the sensitivities similar as done on the [PTDF Dialog](#).

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

Calculate TLR Sensitivities

Click the button to calculate the sensitivities. The calculation processing through each bus and determines the sensitivity of the MW flow of selected device with respect to a change of real power at each bus. When the transactor type is buyer, then the assumption is that power is injected at each bus and absorbed at the transactor object. When the transactor type is seller, then the assumption is that power is injected at each transactor object and absorbed at each bus.

DC Model Options

Click this button to open the [Ref1633003792DC Options](#) page. Here you can adjust the DC options you want used if you are computing the PTDFs with a Lossless DC approximation.

Include only AGCable Generators

If checked, then only generators available for generation control will be included in the TLR sensitivity calculations.

Set Sensitivities At Out-Of-Service Buses Equal to Closest

Click this button to approximate the sensitivities at out-of-service buses with the sensitivity of the closest bus. Otherwise the out-of-service buses will display sensitivities of 0 when the sensitivities are calculated.

Line/XFMR or Interface Only: Bus, Generator, Load, Injection Group, and Area Sensitivities

The Bus Sensitivities show the values calculated at each bus. Generator sensitivities show the same information but have only a list of generators with the P Sensitivity column showing the sensitivity calculated at the terminal bus. The Load sensitivities are similar to the generator sensitivities in that they show the same information as the bus sensitivities but only show the list of loads with the P Sensitivity column giving the sensitivity calculated at the terminal bus. The Injection Group sensitivities represent averages of the bus sensitivities weighted by the participation factors of each generator, load, or switched shunt in the group. All participation factors are included in this calculation regardless of the AGC status of any included generators. The Area Sensitivities calculated represent a weighted average of the generator buses in the area with the weighting done by generator participation factor.

Multiple Elements Only: Multiple Bus, Generators, Loads, and Injection Group Sensitivities

The Multiple Bus Sensitivities show the values calculated at each bus for each device examined during the analysis. Multiple Generator and Load Sensitivities show the same information but have only a list of either generators or loads with the P Sensitivity column showing the sensitivity calculated at the terminal bus. The Multiple Injection Group Sensitivities represent averages of multiple bus sensitivities weighted by the participation factor of each generator, load, or switched shunt in the group. All participation factors are included in this calculation regardless of the AGC status of any included generator.

The results of the multiple element analysis can be accessed using script commands or SimAuto by using the variable MULTBUSTLRSENS with the appropriate data object: Bus, Gen, Load, or InjectionGroup. There will be as many variables with the MULTBUSTLRSENS name and appropriate location number as there are elements that were studied. The location numbers start at 0. The results are assigned in the order that the elements are included in the list of branches or interfaces

to be studied. For example, if the list of branches includes five branches defined in the case in the following order: 1 to 2, 2 to 4, 4 to 3, 3 to 1, 2 to 3, and branches 2 to 4 and 4 to 3 are selected for inclusion in the multiple element TLR calculation, MULTBUSTLRSENS:0 will correspond to branch 2 to 4 and MULTBUSTLRSENS:1 will correspond to branch 4 to 3 for all sets of results. When selecting devices to study based on them being overloaded or contingency-overloaded, the results will appear in the order that the overloaded elements are defined with the case. To determine which elements are overloaded in the base case, use the LINELIMITPERCENT:2 field with Branch data objects and the FGPERCENT field with Interface data objects. If these fields are greater than 100 percent then the element is overloaded. The CTGVIOL variable can be used with the Branch or Interface data type to determine if an element is overloaded due to a contingency. This field will be non-zero for any element that has overloads due to contingencies.

TLR Multiple Device Type

The Multiple Device Type selector allows you to specify a group of elements for which Transmission Loading Relief (TLR) sensitivities will be calculated. The Multiple Device Type selection is accessible only when you chose the Multiple Elements option from the Device Type in the [TLR Sensitivity Dialog](#).

Selected Devices

Select Lines/XFMRs

Select the Selected Devices option, and click on this button to open a dialog to choose the lines and transformers to be included in the calculation. Toggle the Selected field in the Lines and Transformer [Case Information Display](#) to Yes for the branches you wish to include. If the Selected field is not available in the information display, add that column to the display by right-clicking in the display and selecting [Display/Column Options](#). The ETLR of selected Lines/XFMRs is the algebraic sum of the TLRs of each individual element. The WTLR uses as weight the current MW flow in the element.

Select Interfaces

Select the Selected Devices option, and click on this button to open a dialog to choose the interfaces to be included in the calculation. Toggle the Selected field in the interfaces [Case Information Display](#) to Yes for the interfaces you wish to include. If the Selected field is not available in the information display, add that column to the display by right-clicking in the display and selecting [Display/Column Options](#). The ETLR of the selected interfaces is the sum of the TLRs of the interfaces that have a limit different from zero. The WTLR uses as weight the MW flow in the interface.

Overloaded Devices

Select this option to include in the multiple element TLR calculation the branches and/or interfaces that are overloaded in the present case based on the [Limit Monitoring Settings and Limit Violations Dialog](#). The list of the overloaded branches and interfaces is available in the [Limit Monitoring Settings and Limit Violations](#) information display.

CTG Overloaded Devices

Select this option to include in the multiple element TLR calculation the branches and/or interfaces that were identified as overloaded by the [Contingency Analysis](#) tool. A branch or interface is included in the calculation if it has been overloaded during at least one contingency. The list of branches and interfaces identified as overloaded can be accessed from the [Lines, Buses and Interfaces Tab](#) in the [Contingency Analysis Dialog](#).

The ETLR of the CTG overloaded lines and transformers is the algebraic sum of the individual TLRs. The WTLR uses as weight the Aggregate MVA Overload of each transmission line and transformer, which is defined as the sum of the MVA overload in the line or transformer across the contingencies that caused a violation in that particular line or transformer. The Aggregate MVA Overload and a related field, the Aggregate Percent Overload of a line or transformer, are measures of the weakness of that transmission line on the grid. The Aggregated MVA Overload and the Aggregate Percent Overload are line and transformer fields that can be displayed in the [Case Information Display](#), used in [Contouring](#), etc.

The ETLR of the CTG overloaded interfaces is the algebraic sum of the individual TLRs. The WTLR uses as weight the Aggregate MW Overload of each interface, which is defined as the sum of the MW overload of the interface across the contingencies that caused a violation in that interface. The Aggregate MW Overload and a related field, the Aggregate Percent Overload of an interface are measures of the weakness of the interface. These two interface fields can be displayed in the [Case Information Display](#), used in [Contouring](#), etc.

Effective Transmission Loading Relief (ETLR)

In the same manner as the TLR represents the MW increase in an element per MW transfer, the ETLR column in the Mult. Bus Sensitivity table represents the total MW increase in all the elements in the set per MW increase of the transaction. Let us suppose that the set contains two transmission lines: A and B and assume that for a 1 MW transfer from bus i to the transactor, the flow in line A increases in 0.5 MW and the flow in line B decreases in -0.3. Then the ETLR of bus i is +0.2 since that is the total MW increase in the elements in the set. The ETLR provides a measure of the simultaneous MW change in multiple elements, and thus the overall effect in flows on the element of the set. The ETLR is a bus field that can be included in the bus [Case Information Display](#), used in [Contouring](#), etc.

Weighted Transmission Loading Relief (WTLR)

The WTLR column in the Mult. Bus Sensitivity table weights the sensitivities based on the flow or overload flow values of the element. It is a measure of the value of a certain bus to relief transmission loading. The buses with the highest WTLR (or lowest WTLR, depending on the transactor type) are identified as the most effective buses to mitigate transmission loading considering multiple elements. The WTLR is a bus field that can be included in the bus [Case Information Display](#), used in [Contouring](#), etc.

Generation Shift Factor Sensitivities

Generation Shift Factor (GSF) Sensitivities are a specific kind of TLR calculation. GSFs always involve a transfer with the slack bus being the Buyer. Other than this, GSF and TLR calculations are *identical*. See [TLR Sensitivities](#) for more information.

Chapter 11: Time Step Simulation

This chapter contains information on the new Time Step Simulation tool. This tool allows you to set up multiple solution runs with changing input data. This can be useful for running several OPF or SCOPF solutions in sequence, and storing information from each solution in tabular format.

This tool can also be handy as a training tool as well. As an instructor, you can set up a scenarios that the student can interact with, and have changes occur at prescribed intervals during their interaction.

- Time Step Simulation Overview
- Setup and Control
- Schedules
- Running the Simulation

Time Step Simulation

The Time Step Simulation tool allows you to specify operating conditions and obtain power flow solutions for a set of points in time. It provides the tools needed to analyze the operation of a power system hour by hour.

Time Step Simulation is available in the base Simulator package. If you own Simulator, you can start taking advantage of this valuable tool right away. In addition, if you own Simulator OPF or SCOPF licenses, you can solve hourly OPF and SCOPF scenarios and use the tool to evaluate the behavior of prices and operating constraints. The tool will obtain the optimized generation dispatch for each hour of the analysis horizon.

In order to access the Time Step Simulation, go to the [Tools](#) ribbon tab, and choose **Time Step Simulation** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#). Please continue reading the following topics and take full advantage of this powerful tool.

[Time Step Simulation Quick Start](#)

[Time Step Simulation Dialog](#)

[Specifying and Maintaining a List of Time Points](#)

[Loading Hourly Input Data](#)

[Setting up Scheduled Input Data](#)

[Storing Input Data and Results](#)

[Running a Timed Simulation](#)

[Time Step Simulation Toolbar](#)

[Running OPF and SCOPF Time Step Simulations](#)

Time Step Simulation: Quick Start

The first time you access the Time Step Simulation by selecting **Time Step Simulation** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab, you will see the [Time Step Simulation Dialog](#), which contains several pages. The **Hourly Summary** page is used to define and control the time points you want to analyze. As an example, we will assume that you have hourly load data for tomorrow and that you want to determine the system bus voltages for each hour. You would do the following:

Step One: Set the List of Points

In order to create a list of points, right click on the **Hourly Summary** page and select **Insert New Timepoint(s)**, which brings up the [New Timepoint Dialog](#). In this dialog, select tomorrow's date from the drop down calendar component. Set the field **Total Number of Timepoints to Enter** to 24. Assume the other default values and click OK. This will insert 24 timepoints one for each hour starting tomorrow at 1: 00 AM.

Step Two: Specify Input Data

Input data is specified in the **Input Page**. In this example your input data corresponds to hourly loads. Select the **Hourly MW Loads** sub-page. In order to specify hourly load values you have to insert a column for each load. Right Click on the grid and select **Scale/Insert Load Column(s)** to bring up the [Insert/Scale Column Dialog](#). In the selector component, select the load for which you want to specify hourly values. You can press the shift key to select multiple elements. Then press the blue **Arrow Button** to pass the selected loads to the right side of the selector. Select **OK** to insert the new column(s). Now you can specify the hourly MW values for those loads.

Step Three: Specify the Custom Results

Obtaining solutions for a large number of timepoints has the potential to create unnecessary burden in memory and storage due to the large amount of data that can be generated. For this reason, the Time Step Simulation Tool allows you to explicitly specify what quantities you want to display and store. This is done in the [Results Page](#). This page contains grids for many devices including Generators, Lines, etc. In this example we want to analyze the bus voltage magnitudes. We specify what quantities we want to store as results for each object by clicking the **View/Modify** button, which brings up the [Custom Results Selection Dialog](#). In this dialog select the **Buses** page and in the **Available Bus Fields** section check the **Per Unit Voltage** field. Results will be saved only for those buses that have the **Time Selected** field on the grid set to YES. Set this field to YES for the buses that you want to save. Click the **Save and Close** button to save the custom results settings, i.e., which objects, fields and records are to be kept during the solution; in this case bus voltages.

Step Four: Run the Simulation

The upper part of the Time Step Simulation form contains buttons used to control the simulation. To do a full run of the 24 hours click the **Do Run** button. The **Last Result** box shows the progress of the simulation as each time point is being solved. If you are in the **Results – Buses** page, you will see that a column was added for each bus set to YES in the **Custom Results Selection Dialog**. Each column shows the bus per unit voltage. Recall that you can **Right Click** on any Simulator grid and select **Plot Column** to obtain a plot of the column values. If you select cells spanning all the bus per unit voltage columns, you will obtain the voltage profiles for each bus, versus time.

If you are in the **Hourly Summary** page, the **Processed** column shows that each point was in fact processed and solved.

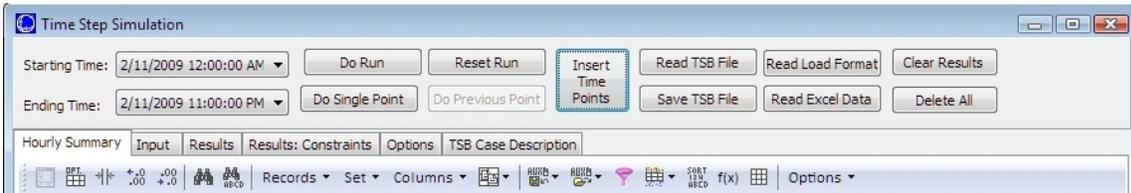
Step Five: Save the Results

Once you have completed the simulation, you can save the results in a **Time Series Binary** (tsb) file by pressing the **Save Data Binary** button. This file will contain all the input data, the simulation

options, the custom results settings, and the results. You can reload this tsb file any time by pressing the **Read Data Binary** button.

Time Step Simulation Dialog

The Time Step Simulation Dialog is used to control and visualize the time simulation. The top section of the form contains buttons for data input/output and buttons to control the progress of the simulation. The main part of the form has a number of [Time Step Simulation Pages](#) that contain grids where input data can be specified and simulation results can be examined.



Input/Output Buttons

Insert Time Points

Press this button to open the New Timepoint Dialog that allows specification of timepoint starting date and time, number of timepoints to enter, and the interval between timepoints.

Read TSB File

Press this button to read a Time Series Binary file (.tsb File). This files stores hourly and scheduled input data, the simulation options and the hourly results.

Read Load Format

Reads a .csv file containing hourly load MW and MVar values. The CSV format of this file is:

```
BUS_NUMBER,BUS_NAME,VST bus numbers,DPID,YrMoDay,Hr,KW,KV
446,ANAME,18803,506520,20041001,1,1970,769
446,ANAME,18803,506520,20041001,2,2005,821
```

Save TSB File

Press this button to save the hourly and scheduled input data, the simulation options, and the hourly results in a Time Series Binary file (tsb file).

Read Excel Data

Reads hourly load and generation data from Excel: The formats for generator and load data are:

Generators:

```
GEN DATA
Date Hour BusNameGen1 BusNameGen2 ..
Bus Numbers 19306 100370 ..
(s)
10/01/05 1 522 340 ..
```


the **Do Run** button caption changes to **Continue Solution**. Press this button to continue the solution.

Do Single Point

Use this button to solve the next time point. You can see the last processed point in the **Last Result** box. You can also select a specific starting point by using the **Start Time Point Selector**.

Reset Run

Use this button to go back to the first time point and initialize the simulation. This action does not delete the results of the time points processed so far, but it resets all the processed fields to No.

The screenshot shows a horizontal control bar with three main elements: a text field on the left containing 'Last Result: Initialized', a text field in the middle labeled 'Present Time', and a checkbox on the right labeled 'Refresh All Displays Each Time Step'.

Last Result

The last result box in the lower left corner of the dialog is used to show solution progress messages. The messages indicate correct solutions or errors in the solution of the particular time point. It also shows the progress of the contingency analysis during a SCOPF solution.

Present Time

If you are solving the timepoints using a [scaled time delay](#), the Present Time of the process is displayed in this field.

Refresh All Displays Each Time Step

Check this box to have all displays refresh after each time step. This could slow the process down if there are a number of displays open.

Time Step Simulation Toolbar

The purpose of this toolbar (besides providing shortcuts) is to command a Time Step Simulation without having to keep the [Time Step Simulation Dialog](#) open. This is particularly important during a **Timed Simulation**, in which you want to see the changes on the oneline diagram as they occur in time.



The Figure shows the main functions of the toolbar. [Show/Hide the Time Simulation Form; Play/Pause/Reset/Next Time Control Buttons; Last Result Box; Present Time; Progress Bar, and Timed Simulation Options]. All the buttons, except the Show/Hide Time Simulation Form are disabled until a list of time points have been defined in the [Hourly Summary Page](#). The **Time Step Simulation Toolbar** is available in **Run Mode** and it is visible only when the [Time Step Simulation Dialog](#) is open.

Time Step Simulation Pages

The Time Step Simulation Dialog contains a number of pages used to specify hourly and scheduled input data, and to examine the results of the simulation. Please continue reading the following sections for a detailed explanation of each page:

[Hourly Summary Page](#)

[Input Page](#)

[Results Page](#)

[Results: Constraints Page](#)

[Options Page](#)

[TSB Description Page](#)

Hourly Summary Page

The Hourly Summary Page of the [Time Step Simulation Dialog](#) is used to define the time points and to display a summary of the simulation. Most of the commands to manage time points can be accessed from the [Hourly Summary Page Local Menu](#).

Once timepoints are defined, the Summary Page presents the following columns for each time point:

Date

The date of the time point. To modify the date, Right Click and select **Change Timepoint Time** on the **Local Menu**.

Hour

The time of the time point to the minute. To modify the time, Right Click and select **Change Timepoint Time**.

Skip

This field is set to YES to include the time point in the Simulation. If set to NO, the hourly input data and any schedule action that occurs at this time point is not to the power system.

Processed

This field is set to yes if the simulation has processed the time point. This includes applying the hourly input data, applying the scheduled actions and solving the power flow for that particular time point.

Solution Type

The time point can be solved using one of the following **Solution Types**: Single Solution, Unconstrained OPF, optimal power flow (OPF), and security-constrained optimal power flow (SCOPF). The last three solution types require the Simulator OPF/SCOPF add-ons.

Run Contingencies

If the case has a defined set of [contingencies](#), you can choose to run the contingency analysis for each time point by setting this option to YES.

Solved

YES if a solution was obtained for the timepoint under the specified solution type.

Num Loads

Total number of loads for which hourly input data has been specified.

Total MW Load

Total MW load specified as hourly input data for the timepoint

Total Mvar Load

Total reactive load specified as hourly input data for the timepoint

Num Gens

Total number of generators in the system for which hourly input data has been specified

Total MW Gen

Total MW of generation specified in the hourly input data.

Pre Script Cmd

The Time Step Simulation has the capability of running a pre script command before each time point. The pre script command is run right before the time point is solved. Please check the [Script Command](#) section to learn the details about performing Simulator actions using script commands.

Post Script Cmd

The Time Step Simulation has the capability of running a post script command right after the time point is solved. Please check the [Script Command](#) section to learn the details about performing Simulator actions using the script language.

Hourly Summary Page: Local Menu

The Hourly Summary Page Local Menu is used to perform a number of logical actions on the timepoint grid and is, as every Simulator local menu, accessed by right clicking anywhere on the grid. When selected from the Hourly Summary Page, the local menu shows the following additional time step simulation specific options:

Apply Time Point

Simulator applies the hourly input data of that particular time point to the power system model, without solving the time point. Upon selection of this option, the timepoint information can be visualized in the [Case Information Displays](#).

Solve Time Point

This option applies the time point hourly and the scheduled input data to the power system, and solves the timepoint after the specified solution type.

Change Time Point Time

This option allows the user to modify the date time of the selected timepoint. If the new date time belongs to an existing point in the list, an error message is displayed. The timepoints can be specified with an accuracy of up to one minute.

Display/Column Options

Select this option to add or remove columns to the hourly summary grid.

Set/Toggle/Columns > Select Column(s)

This option is normally used to select the entire column. You can drag the mouse across several columns in a row, and use this option to select multiple columns. Then you can for instance copy the entire column data to the clipboard, or plot all the selected columns.

Set/Toggle/Columns > Select Row(s)

This options is normally used to select an entire row. You can drag the mouse across several rows within a column, and use this option to select multiple rows. Once the rows are selected, you can perform actions with the selected rows, such as copying data to the clipboard.

Set/Toggle/Columns > Contour Column

Use this option to contour a column in the data grid.

Insert New Timepoint(s)

Use this option to insert new timepoints into the timepoint list. This selection brings up the [New Timepoint Dialog](#).

Set/Toggle/Columns > Plot Column(s)

Use this option to automatically generate column plots. By default, in the Time Step Simulation grids, the column plots display graphs of column data versus the datetime column (the combination of the date and hour columns).

Delete Entire Timepoint Record

Use this option to delete the selected timepoint and all the hourly input data and custom results of that timepoint.

Input Page

The Time Step Simulation Input Page is used to specify hourly input data and scheduled data. This requires that the list of timepoints have been created. Please read the [Specifying and Maintaining a List of Timepoints](#) for details on managing the list of time points.

The Input Page contains several pages that can be grouped in two types: Hourly Input Pages and Schedule Pages.

Hourly Input Pages

The hourly input pages (Hourly MW Loads, Hourly Mvar Loads, Hourly Actual MW Generation, Hourly Maximum Generation, Hourly Line Status, Hourly Area Loads, and Hourly Zone Loads) are [Matrix Grids](#) that are used to specify data on an hour by hour manner. In order to tell Simulator that we want to specify hourly data for a load, generator, or line, we need to add that particular object to the corresponding grid. For instance, suppose that you want to specify hourly MW data for Load 1 at bus 1. Then go to the Hourly MW Loads page, right click and select **Insert/Scale Load Column(s)** to bring up the [Insert/Scale Column Dialog](#). In this dialog, you can select the load and add it as a column to the Load grid. The corresponding values for that particular load can then be entered.

A similar process is followed to add one or multiple hourly MW generation outputs or hourly line statuses.

Use the Hourly Area Loads and Hourly Zone Loads pages to specify the hourly MW load for an entire Area or Zone. When Simulator applies the input data to solve a timepoint, it will scale the load of the specified area or zone to match the value entered for that hour.

Schedule Pages

The schedule pages ([Schedules Page](#) and [Sched Subscriptions Page](#)) are used to specify scheduled input data, i.e., data that does not follow an hour by hour format. Examples of data that can be scheduled are scheduled transactions, generator statuses, line statuses, etc. For a detailed explanation on how to set up scheduled data, please read the [Setting Up Scheduled Input Data](#) section.

Matrix Grids

Matrix Grids are a special type of data grids used in the input and results pages of the Time Step Simulation tool. In these grids, the time dimension is assigned to the rows and the object fields (load MW, generator MW output, etc) are assigned to the columns. The column header corresponds to the ID of the object. The number of columns of the grid depends on the user options and simulation results:

Hourly Input Pages

In the case of input pages, the user has to specify hourly data for each object, e.g., load, generator, area. Thus the user adds each column to the grid explicitly.

Custom Result Pages

In the custom result pages, a column is created for each object whose results have been specified to be stored.

Constraint Pages

Since the constraints are determined at solution time, the columns appear only when constraints have been detected during the solution.

Results: Constraints Page

This page is only available in OPF and SCOPF versions of Simulator. When the OPF solution type is used to solve a timepoint, the status of the system during that particular hour is optimized so that the total operating cost of the system is minimized and the normal operation constraints are enforced. In addition, the SCOPF solution type enforces contingency constraints. The OPF and SCOPF solutions contain information about the elements that determine the LMPs, the binding constraints, the violating contingencies, and changes in the control settings.

The information related to contingencies on these pages is available only in the SCOPF Simulator add-on.

The pages of the **Results: Constraints Page** are:

Results Summary

This page is similar to the [Hourly Summary Page](#), except that it contains additional operating information such as:

Initial Cost: The cost of operating the system given the initial generator set points and a standard power flow solution. No controls are moved to minimize cost

Unconstrained Cost: Is the total operating cost after the controls are moved to minimize cost, without enforcing any normal operation or contingency constraint. The result is an operating cost that corresponds to an Economic Dispatch solution.

Unconstrained LMP: Is the average marginal price of the system under unconstrained optimization.

Final Cost: Is the total operating cost after a constrained (OPF or SCOPF) solution has been obtained.

LMP (Average, Standard Deviation, Minimum and Maximum): Metrics of the LMP values.

Binding Lines: Number of transmission lines and transformers that are binding after the OPF/SCOPF solution has been determined.

CTGs Unsolvable: Number of unsolvable contingencies. These are severe contingencies that would cause the power flow solution to fail for that particular time point scenario. Unsolvability of the power flow case is related to maximum loadability conditions.

Hourly Binding Lines

This is a [Matrix Grid](#) that shows information for transmission lines and transformers that become binding constraints during the OPF or SCOPF solutions. A similar page is available for binding interfaces.

When transmission lines or transformer thermal ratings become binding constraints for a timepoint, a column is automatically added forming in this manner the matrix grid. You can access the details of the binding line or transformer by right clicking and selecting Show [Binding Constraint Dialog](#) on the **Local Menu**. The grid shows also the following summary columns.

Processed: YES if the time point was correctly processed.

CTGs with Viols: Number of contingencies that presented one or more violations (violating contingencies)

CTGs Unsolvable: Number of unsolvable contingencies for the timepoint

BC Line Viols: Number of transmission line and transformer violations that were identified in the base case.

Line Viol: Number of transmission line and transformer thermal violations.

Binding Lines: Number of binding lines in the OPF/SCOPF solution.

Line Unenforceable: Number of lines with unenforceable limits.

Hourly Binding Interfaces

This is a [Matrix Grid](#) that shows information for interfaces that become binding constraints during the OPF or SCOPF solutions. A similar page for lines is available.

When interfaces become binding constraints for a timepoint, a column is automatically added to the matrix grid. You can access the details of the binding interface by right clicking and selecting [Show Binding Constraint Dialog](#) on the **Local Menu**. The grid shows also the following summary columns.

Processed: YES if the time point was correctly processed.

CTGs with Viols: Number of contingencies that presented violations (violating contingencies)

CTGs Unsolvable: Number of unsolvable contingencies for the timepoint

BC Interface Viols: Number of interface violations that were identified in the base case.

Interface Viol: Number of interface violations.

Binding Interface: Number of binding interfaces in the OPF/SCOPF solution

Interface Unenforceable: Number of interfaces with unenforceable limits.

Hourly Binding Contingencies

This [Matrix Grid](#) shows information similar to that found on the Hourly Binding Lines and Hourly Binding Interfaces pages. Here though the data is organized by contingencies, which allows easy identification of the most severe contingencies. Each column of the matrix grid corresponds to a contingency. The grid shows also the following summary columns.

Processed: YES if the time point was correctly processed.

CTGs with Viols: Number of contingencies that presented violations (violating contingencies)

Line Viol: Number of transmission line and transformer thermal violations.

Binding Lines: Number of binding lines in the OPF/SCOPF solution

Binding Interfaces: Number of binding interfaces in the OPF/SCOPF solution

Base Case: Shows the number of violating contingencies in the base case.

Binding Line Summary Matrix

This [Matrix Grid](#) shows the number of hours that a line has been binding due to each contingency. In this case, the rows correspond to binding lines, and the columns to violating contingencies. You can access the details of the binding element by right clicking and selecting [Show Binding Constraint Dialog](#) on the **Local Menu**. The grid shows also the following summary columns

From Number: Binding line from bus number

To Number: Binding line to bus number

Circuit: Binding line circuit ID

Total Hrs: Total number of hours the line was binding (in the overall simulation).

Total Hrs Unenforceable: Total number of hours the line constraint was unenforceable (in the overall simulation)

Basecase: Number of hours the line was binding in base case solutions.

Binding Line Summary List

Shows information similar to the **Binding Line Summary Matrix** but in form of a List. The same line may appear several times under different contingencies. Besides the information on the **Binding Line Summary Matrix**, this grid shows the following fields.

Contingency Name: Contingency that causes the line constraint to be binding

Avg MC: Time Average MVA Marginal Cost

Max MC: Time Maximum MVA Marginal Cost

Min MC: Time Minimum MVA Marginal Cost

Binding Elements Dialog

This dialog is called from the [Results: Constraints Page](#) of the [Time Step Simulation](#) dialog. Right-click on a time point record in the Results: Constraints table, and select **Show Dialog** from the popup menu. The dialog shows the binding line and interface constraints determined by the OPF/SCOPF solution. The **Time** selector allows easy navigation through the list of time points.

The grid section of the dialog shows the following:

Type: Either line or interface flow

Constraint ID: Line or Interface ID

Contingency Name: Name of the contingency under which the constraint becomes binding.

MVA Marg. Cost.: Marginal cost of enforcing the constraint. For lines, the limit corresponds to the thermal (MVA) limit. For interfaces, it is given by a MW limit.

Results Page

The results of the Time Step Simulation are presented on the grids of the **Results Page**. The Time Step Simulation allows you to specify what objects (buses, lines, generators, loads, etc) and what object fields (bus voltage, bus LMP, gen MW, etc) should be displayed on the result grids. This gives the user the flexibility needed to explore the relevant results, avoiding at the same time the problem of storing a massive amount of results, most of which may not be relevant. Storage is a critical aspect of the Time Step Simulation, since a set of results comparable to full a PF/OPF/SCOPF solution is generated for each timepoint.

The **Results Page** has two sections: The top section is used to set up the options needed to customize the results display. The grid section is used to display the actual results.

Results Page: Top Section

This section includes the following options:

View/Modify

Press this button to access the [Custom Results Selection Dialog](#). This dialog is used to specify the objects and object fields for which results will be stored.

Load

The custom result definitions set up in the [Custom Results Selection Dialog](#) can be saved in a **Results File** in order to use them with different **tsb files** or Simulator cases. Press this button to Load the Results File and apply the result definitions to the current Time Step Simulation.

Save

Press this button to save the result definitions in a **Results File**.

Group Results by

The Result Grids for Areas, Buses, etc are [Matrix Grids](#) that present the hourly results for each type of object. For instance, suppose that we want to store the bus voltage magnitude and angle. The Buses grid will show columns for the hourly values of voltage magnitude and voltage angle of each bus.

Objects: The columns will be grouped by objects, e.g., all the fields of bus 1, then all the fields of bus 2, etc.

Fields: The columns will be grouped by fields, e.g. all the bus p.u. voltages, then all the bus LMPs, etc.

Identify Results by

The Result Grids for Areas, Buses, etc. create one column for each object field specified in the [Custom Results Selection Dialog](#). The column header thus identifies the particular object for which hourly results are displayed. This identification can be made based on the object **Number**, **Name** or **Number + Name** combination.

Results Page: Grid Section

This grid section is used to display the results of the [Custom Results Grid Pages](#).

Custom Results Selection Dialog

This dialog is used to specify what objects and object fields will be stored during the Time Step Simulation and will be displayed on the [Results Grid Pages](#). This dialog is opened using the **View/Modify** button on the [Results](#) page of the Time Step Simulation dialog.

In order to tell the Time Step Simulation that you want to store a particular field for a certain object, you need to:

1. Click on the page of the object type you need to store (Area, Bus, etc)
2. Set to YES the **Time Selected** field of those objects for which you want to store information.
3. Check the boxes corresponding to those fields you want to store (bus LMP, bus p.u. voltage, etc)

Once you are done with the selections, press the **Save and Close** button to apply the customization. The [Custom Results Grid Pages](#) will be filled with the corresponding columns after the Time Step Simulation starts.

Note that the actual results and the result customization will be stored in the tsb file. For more information about saving the Time Step Simulation results, please read the [Storing Input Data and Results](#) section.

The following object's fields can be specified to be displayed on the grids and stored in the tsb file.

Areas

Buses

Generators

Injection Groups

Interfaces

Lines

Owners

SuperAreas

Transformers

Zones

To choose the fields for a specific type of object, click on the **Add/Remove Fields** button and select the fields from the list of available fields on the left side of the **Select Fields** dialog, and move them to the **Selected Fields** list on the right side of the dialog. Click OK to choose the selected fields.

Results Grid Pages

These are the Pages of the [Results Page](#) that show the results of the Time Step Simulation after the results definitions specified in the [Custom Results Selection Dialog](#).

There is one page for each type of object: Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Owner, Superareas, Transformers and Zones.

These pages are all [Matrix Grids](#). Each column of the grid corresponds to a field of a specific object of the power system, e.g., KV voltage of bus 1. Each row of the grid corresponds to a timepoint.

Note that you can plot the results of the grid versus the timepoint date time by selectiing **Plot Column(s)** on the **Local Menu**.

Time Step Simulation Options

The Time Step Simulation is controlled through a set of options specified in the **Options Page**. The **Result Options** and the **Pricing Options** are available on this page only in the OPF/SCOPF Simulator add-ons.

Area and Zone Load Scaling

In many practical studies, hourly data of individual MW load may not be available for all the loads in a control area. In order to simulate load variations, Simulator allows you to specify the hourly Area or Zone Total MW Load values in the **Hourly Area Loads** and Hourly **Zone Loads page** of the [Input Page](#). Thus the Time Step Simulation load data may be a combination of:

- Areas or zones where each individual load is specified
- Areas or zones where only the total hourly MW load is known.
- Areas or zones where some individual loads and the total area load are known.

If you have specified individual load values for a load within an area or zone that is also set to be scaled to a total MW value, Simulator will adjust the individual load to the individual value first, and then the load will be scaled with the rest of the loads in the area or zone to achieve the total MW value desired for the hour. The final load will be the total value specified for the area or zone.

When hourly values are specified for areas and zones within the same time step input, only one or the other can be used. This can be set in the **Use Hourly Values of** option.

You can also specify in the **Reactive Power Scaling** option how the reactive power is treated when the load is scaled. You can choose to have the reactive power scaled to keep the power factor constant, or for the reactive power to remain fixed at its original value.

Solution Options

These options control the solution process.

Pause if Power Flow Does Not Solve: The Time Step Simulation is stopped at the timepoint where a power flow solution cannot be obtained. This is an indication of wrong data or a system brought to its loadability or transfer capability limit.

Enable Power Flow Area Interchange: This option ensures that the area interchange control, if possible, is enforced in the case.

Use Parallel Contingency Analysis (on Dual CPU Machines): Use this option to speed up the computation of the contingency analysis by distributing the processing time on several processors. This option is available for SCOPF.

Pricing Options

These options are available only in the OPF/SCOPF add on.

Solve Unconstrained Case: Select this option when you want an unconstrained solution to be obtained before an OPF or SCOPF solution for each time point. When combined with the **Results Options -> Save Unconstrained Generator MW Outputs**, the MW Generator outputs obtained in the unconstrained solutions are saved.

Price Hydro Generation at Marginal Cost: During OPF and SCOPF simulation, hydro generation may experience large changes in output due to its low marginal cost. However, hydro generation is often not as cheap if limited water levels and dam restrictions are observed. These considerations are usually taken care of in the hydro-thermal coordination solution, outside of Simulator. In the OPF and SCOPF solutions it is important to assign a reasonable price to hydro generation to avoid large

generation output deviations. A common mechanism to do that is to first obtain the system marginal cost, and then assign this cost to the hydro units.

Reset Hydro Generation Price at the End of Time Period: Choose this option to make the hydro generation price be reset for the next time point solution.

Save Binding Constraints: The binding constraints determined in the OPF/SCOPF are stored.

Time Step Simulation Options

The Time Step Simulation can be performed in two ways:

Continuous: In this case Simulator solves one time point after another immediately. The purpose of the simulation is to obtain the solutions for all time points as quickly as possible.

Timed: The purpose is to simulate the solutions as they would occur in actual time. In this case, the difference in date time between two time points in the list defines a delay to start the solution of the second point. A **Time Scale** is used to set the speed of the simulation with respect to actual time. Suppose that you have 3 time points defined at the following date times:

1/20/05 1:00 AM

1/20/05 2:00 AM

1/20/05 4:00 AM

Assume also that the **Time Scale** is 1 hour runs in 10 seconds. If you start the **Timed Simulation** you would see the conditions of the first time point applied to the power system immediately, the conditions of the second time point applied 10 seconds later, and those of the third point applied 20 seconds after the second point. The delays on the simulation allow you to see how the quantities evolve in actual time. In addition, you can animate the time simulation while each time point is being solved. The visualization of the **Timed Simulation** is enhanced when you use the [Time Step Simulation Toolbar](#).

Step Type: This section of options allows you to specify if the data for each time point should be applied only, or if the data should be applied and the load flow solved. You can also specify if the specified hourly data in the Input page should be applied, if schedules should be applied, and if the solution of the previous time step should be applied as the starting point for the next time step.

Auto Contouring Options

The Time Step Simulation allows you to contour quantities on the oneline diagram using Simulator [Contouring](#) at each timepoint. Optionally, these contour diagrams can be saved in different formats.

No Auto Contouring: Contouring is not used during the Time Step Simulation. Although using the **Timed Simulation** you can see the quantities change and the animation take place on the diagrams, the contouring is not displayed.

Contour but Do not Save: Contouring takes place at each timepoint, but the diagrams are not saved.

Save in File as Bitmap: The contouring diagrams generated at each timepoint are saved in Bitmap format.

Save in File as JPEG: The contouring diagrams generated at each timepoint are saved in JPEG format.

Contour File Name Format: The Bitmap or JPEG contouring diagrams are saved using the specified format, which includes the timepoint date time.

Auto Load TSB File Options (saved in case pwb file)

These are options that relate the power system case (.pwb File) to the time series binary file (.tsb file). These options are saved with the .pwb case.

Automatically Load Default *.tsb File: The tsb file specified in the Default *.tsb file is loaded automatically when opening the .pwb case. If the file cannot be found, a message will be issued.

Automatically Run Simulation after Loading *.tsb: This option is only relevant if the tsb file was automatically loaded, and instructs the TSS form to solve the load flow immediately following loading the tsb file.

Automatically Set Default *.tsb File to Current *.tsb File: When leaving Simulator, the current time series information is saved in the Default *.tsb file.

Default *.tsb File: Default path and name of the *.tsb file.

Run Pre-Script Command

This option allows you to choose if script actions specified to be run before solving the load flow for certain hours should be implemented before or after specific input data designated for that hour is applied.

Specifying and Maintaining a List of Timepoints

The list of timepoints is the basis for the Time Step Simulation. Simulator will go through the list of timepoints and solve each one of them in sequence. Results will be available only for those timepoints specified in the list.

Whether you start with an empty list or you already have time points in it, you can insert new time points by Right-Clicking in the **Hourly Summary** page and selecting **Insert New Timepoint(s)**. This will bring up the [New Time Point Dialog](#).

The list of timepoints will be always sorted based on the date time shown on the date and hour column in the [Hourly Summary Page](#). Thus, if you create a new time point with an intermediate date time, Simulator will insert it at the appropriate place in the list. If you need to change the date time of a timepoint you can **Right-Click** in the **Hourly Summary** grid and select **Change Timepoint Time** to bring up the [Change Timepoint Time Dialog](#). This dialog is similar to the [New Timepoint Dialog](#), with the exception that instead of specifying the date time for a new timepoint, you will be modifying the date time of an existing time point. If the date time matches the date time of an existing time point, a warning message is issued.

Each timepoint is linked to its hourly data in what is called a **Timepoint Record**. The **Timepoint Record** contains the date and time of the time point, all the input hourly data specified for that point, and if any, the results that have been obtained for that timepoint. If you delete a timepoint, the entire record is deleted with it. In order to delete the timepoint you can **Right Click** in the **Hourly Summary** grid and select **Delete Entire Timepoint Record**. Note that you can vertically select cells in this grid and delete several timepoints records at a time.

Suppose that you have a 1:00 AM and a 3:00 AM timepoints that have associated with them hourly MW load data. If you insert a 2:00 AM data, the 2:00 AM cell of the MW load will appear empty. If you run a study, no data will be applied to the power system at 2:00 AM, but you will get a result (identical to the one of 1:00 AM). You need to fill the 2:00 AM cells with data in order for the values to be applied at that timepoint and obtain the correct results for 2:00 AM.

New Timepoint Dialog

The New Timepoint Dialog is used to define new timepoints. One or multiple timepoints can be defined at once. Use the date drop-down box to bring up a calendar for easy selection of the date. Note that the calendar has visual controls that allow you to navigate through months or years, which allows you to set the desired date easily.

If this is the first timepoint you will insert in the list, then by default the dialog is populated with today's date at 1:00 AM. If there are other points in the list, the default is one hour after the date time of the last time point.

The other options in this dialog are:

Date

Use this control to select the date of the time point. You can use the visual controls or the up or down arrow keys to modify the date.

Time

Use the control to enter the hour, minute and AM/PM description of the time. Once you are positioned on the hour, minute or AM/PM values, you can use the up/down arrow buttons for easy selection without having to type. Note that the timepoints are specified with an accuracy of up to minutes.

Total Number of Timepoints to Enter

By default this field is set to one, meaning that you will enter only one timepoint with the specified date and time. If the value is more than one, several time points will be inserted in the list at the interval defined in the next two fields.

New Timepoint Interval: Hours

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the hour(s) of the interval between each time point. Timepoints will be Hours + Minutes + Seconds apart. Default is one hour.

New Timepoint Interval: Minutes

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the minute(s) of the interval between each time point. Timepoints will be Hours + Minutes + Seconds apart. Default is zero minutes.

New Timepoint Interval: Seconds

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the second(s) of the interval between each time point. Timepoints will be Hours + Minutes + Seconds apart. Default is zero minutes.

Change Timepoint Time Dialog

The Change Timepoint Time Dialog is used to modify the date time of a time point. Only one time point time can be changed at a time. If the new time assigned to the timepoint already exists, a warning message is generated asking to input a different date time. If the new date time does not exist, Simulator moves the time point to the correct position in the timepoint list.

Date

Use this control to select the date of the time point. Besides the visual controls, you can use the up or down arrow keys to modify the date.

Time

Use the control to enter the hour, minute, seconds and AM/PM description of the time. Once you are position on the hour, minute, seconds or AM/PM value, you can use the up/down buttons for easy selection without having to type. Note that the timepoints are specified with an accuracy of up to seconds.

TSB Case Description Page

This page is used to describe the time series binary file (tsb file) for information purposes. Simulator shows also the **Version Used to Store the TSB File** and the **Simulator Build Date**.

Insert/Scale Column Dialog

The Insert/Scale Column Dialog is called from the [Input Page](#) grids. These [Matrix Grids](#) are used to specify hourly input data for Load MW, Load Mvar, Generator MW, Generator Maximum MW and Area Total MW Load. Hourly data of each particular object is specified in columns. When no column has been added to the grid, the Insert/Scale Column Dialog allows you only to select the new object. On the other hand, when there are existing columns on the grid, the Dialog allows you to insert a **New Column** either alone or based on the values of the existing columns. This option is available because values such as hourly load MW data tend to experience similar fluctuations in time.

Current Column: If there are existing columns and the **Insert/Scale Column Dialog** is called from an hourly input data column, this field shows the position of that column. This field tells the user what column the **New Column** will be based on.

Action:

These options tell how the new column will be inserted.

Scale Entire Current Column: This option is available only when the current column corresponds to an existing hourly data column. When applied, the new column takes the values of the current column scaled by the **Scaling Factor**.

Scale Selected Rows of Current Column: The Time Step Simulation allows you to select a group of contiguous rows of a column and scale only those rows of the **New Column**.

Insert New Column Derived From Current Column: This option copies the values of the **Current Column** to the **New Column**.

Scaling Factor: Factor used to scale the new column.

Load Scaling: These options are available only when inserting Load columns and are used to scale **Real and Reactive Load, Just Real Load, or Just Reactive Load**.

New Columns: Use this selector to specify the ID of the object that will be added as new column to the grid.

Selector: The bottom section of the dialog is a selector that allows you to specify one or multiple objects at a time and add columns for them by passing the objects from the left side to the right side. This selector has the following controls:

Sort by Name: Check this option to sort the list of available objects by name

Sort by Number: Check this option to sort the list of available objects by number

Define Filter: Press this button to filter the list of available objects using an [Advanced Filter](#).

Use Area/Zone Filters: Check this box to filter the list of available objects using the Area/Zone filters.

Search Next: By typing the start of the name of an object in the edit line, you can search the next object or all the objects available which match the search pattern. You can also use wildcards to search for objects.

List of Objects: The objects in the list are selected by clicking on them. Multiple objects that are together can be selected by clicking the mouse while holding the SHIFT key. Multiple objects that are not contiguous in the list can be selected by clicking the mouse while holding the CONTROL key.

Select Button: Press the blue arrow button to pass the selected objects on the left to the right.

Remove Button: Press the trash can button to remove the objects from the selected list on the right side.

For more information about methods to specify hourly input data, please read the [Loading Hourly Input Data](#) section.

Loading Hourly Input Data

The Time Step Simulation allows you to specify the operating conditions of your power system through input data. The input data can be of two types:

Scheduled Data, which is specified for data that is not hourly by nature. To set up schedule data please read the [Setting up Schedule Input Data](#) section.

Hourly Data, or more precisely time point-based data, is specified for each time point in the [Input Pages](#). There are several ways to specify hourly input data:

- By entering data manually on the [Input Pages](#).
- By deriving or scaling values from another column using the [Insert/Scale Column Dialog](#).
- By loading previously formatted data from Excel or .csv files through the **Read Buttons** in the [Time Step Simulation Dialog](#).
- By pasting data from Excel directly to the grid. A common way to do that is to:
 - o Set up the desired timepoints and data columns on the grids of the [Input Pages](#)
 - o Copy the template to Excel by selecting **Copy** on the **Local Menu**
 - o Fill the Excel sheet with the appropriate data
 - o Paste the data back to the data grid by selecting **Paste** on the **Local Menu**

Setting up Scheduled Input Data

There are two types of input data for the Time Step Simulation:

Hourly Data, which is described in the [Loading Hourly Input Data](#) section; and,

Scheduled Data, which is specified for data that is not hourly by nature. In this section we describe how to specify this type of input data.

Although it is possible to specify the operating conditions of a power system exclusively by hourly quantities, there are several quantities that are not hourly by nature, and whose hourly specification would be redundant and would require significant memory storage. Examples of such quantities are:

- The status of a transmission line that is taken out of service on a particular date and time for maintenance.
- The status of a generator, which follows a particular maintenance schedule.
- A generator's voltage set point that is different during the day or at night.
- A scheduled transaction between two areas that has different MW set points applied at 10 am, 4 pm and 10 pm.
- A capacitor connection status for a Mvar block that is used only during the day.
- An industrial load that operates at different levels for different shifts.
- A peaker generating unit that operates only during certain hours of the day

And many others. All these quantities can be specified by introducing the concept of **Schedule**. A schedule is a list of pairs (**Date Time, Value**), where the value can be numerical, conditional (Yes/No or Closed/Open) or text. The Schedule can have any number of time points and can be periodic. The schedule defines the "shape" of how a quantity varies in time. In order to define a schedule, go to the [Schedules Page](#) of the **Input Page**, and select **Insert New Schedule** to bring up the [Schedule Dialog](#). Please follow this link for a detailed explanation on how to define a schedule.

Once a schedule has been created, we can assign an object field, such as the status of a transmission line or the MW output of a generator to the schedule by means of a **Schedule Subscription**. The object field will follow the schedule "shape" in time. The use of Schedule and Schedule Subscription objects gives us great flexibility in specifying how quantities should vary. In particular, it is possible to assign many fields to the same schedule. In order to define a **Schedule Subscription**, go to the [Sched Subscriptions Page](#) of the **Input Page**, and select **Insert New Subscription** to bring up the [Sched Subscription Dialog](#). In this dialog you will encounter a detailed explanation of the Schedule Subscription use and capabilities.

An important feature of the Schedules is that their date times don't need to match the date times of the list of timepoints (time points listed in the Hourly Summary Page). Suppose that your list of time points are defined hourly for the next day: 1am, 2am, etc. up to 11pm. You can schedule a particular action to take place at 2:35 am and see the results of that action in the next time point, i.e., 3 am. A special logic takes care of applying scheduled action at appropriate time points asynchronously.

Schedule Dialog

The Schedule Dialog is called from the [Schedules Page](#) and is used to define a schedule. A schedule is a list of time points together with a numeric, conditional (Yes/No or Closed/Open) or text value that defines the "shape" of how a quantity should vary in time. The timepoints are listed on the grid section of the dialog. The **Schedule Dialog** also contains a number of options that define the schedule.

When a schedule is created, the time step Simulation inserts by default four timepoints for the current date with a numeric value of zero in every point. You can then change the date and times of the schedule timepoints and specify different values.

Schedule Dialog Local Menu Actions

The Schedule Dialog grid, as any other Simulator Grid has a Local Menu that is accessed by right-clicking on any cell. The most important options of this dialog are:

Insert New Point(s): Select this option to bring up the **New Schedule Point Dialog**. This is the same [New Time Point Dialog](#) but is used here to insert Schedule timepoints.

Change Time(s): Select this option if you want to change the date time of a schedule timepoint. The selection brings up the **Change Time Dialog**, which is the same [Change Timepoint Time Dialog](#) used here to change the date time of a schedule time point.

Delete: Use this option to delete the current schedule time point

Select Column(s): Use this option to select an entire column. Note that you can use this option and then delete all the timepoints.

Copy: Use this option to copy the values in the Schedule grid to the clipboard. If you are using Excel, you can modify the values and then Paste back the new schedule points into Simulator.

Timepoint List Options

These options provide shortcuts to some of the **Local Menu** actions related to the maintenance of the list of Schedule time points:

Add Point: Adds a new timepoints at the end of the list. By default the date time of the new timepoint is one hour after the last time point. If there are no timepoints in the list, a timepoint is inserted with today's data at 1:00 AM.

Delete Point: Deletes the selected timepoint(s).

Shift Date Time Buttons: Use these buttons to move by one week, day, hour, minute or second the date time of the selected time point.

Schedule Options

The upper part of the **Schedule Dialog** is used to define other Schedule Options:

Schedule Name: Enter here the name you want to give to the schedule. This is how you will identify the schedule when you set up schedule subscriptions. If the user does not specify otherwise, Simulator assigns names Sched1, Shced2, etc. as new schedules are created.

Name Suffix: This is a string automatically generated by Simulator, which is used to display the main characteristics of the Schedule together with the name. The Name Suffix is composed of three strings separated by points:

[**Num, Y/N or Txt**]: Indicates whether the schedule values are numeric, conditional or text.

[**NPER** or **PERxdyh**]: The string indicates whether the schedule is non-periodic (NPER) or periodic (PER). If it is periodic, it indicates the days (x) and hours (y) of the period.

[**Nn**]: Indicates the number of timepoints (n) of the Schedule.

By looking at the Schedule suffix, the user effectively avoids writing long names for the Schedule to characterize its properties.

Value Type: Numeric, conditional (Yes/No or Closed/Open) or Text. When you toggle this selection, the value column in the Schedule grid changes its heading from Numerical Value to Yes/No value to Text Value, and the numerical cells (blue) are changed to Yes/No cells (green), indicating that you can toggle the cell value, to blank for Text Value. Note that numeric fields, such as Load MW, Schedule Transactions MW etc, will subscribe to a Numeric Schedule, whereas fields such as Line Status will subscribe to a Yes/No Schedule. Text schedules can be subscribed to custom string and memo fields. For more details on Schedule Subscriptions, please read the [Sched Subscriptions Dialog](#).

Periodic Options

A schedule becomes periodic when its values are repeated every certain period specified in days, hours, minutes and/or seconds. A logic condition for a schedule to be periodic is that its period be larger than the time span between the date time of the last time point and the first time point in the schedule. For instance, if the schedule has three time points at 1 am, 2 am and 4 am, the span is 3 hours. Thus the schedule period must be 4 hours or more. If this condition does not hold, Simulator issues a warning message.

Repeat Every: Check this box to make the schedule periodic. The schedule must have a total period greater than 0 to be considered periodic. The schedule period is the number of days plus the number of hours specified.

Days: A positive integer that specifies the number of days of the schedule period.

Hours: A positive integer from 0 to 23 that specifies the number of hours of the schedule period.

Minutes: A positive integer from 0 to 59 that specifies the number of minutes of the scheduled period.

Seconds: A positive integer from 0 to 59 that specifies the number of seconds of the scheduled period.

Valid From: When checked, this option sets a validity date for the schedule, whether it is periodic or not. No schedule actions will be applied to the power system before this date. By default, the Valid From date is set to Jan 01, 2000, at 12:00 am.

Valid Until: When checked, this option sets a validity date for the schedule, whether it is periodic or not. No schedule actions will be applied to the power system after this date. By default, the Valid Until date is set to Dec 31, 2030, at 12:00 am.

Schedules Page

The **Schedules Page** is used to display all the defined schedules and their properties. The most relevant options of **Local Menu** of this page are:

Schedules Page Local Menu Actions

Insert New Schedule: This option is used to define a new schedule through the [Schedule Dialog](#)

Delete: Use this option to delete the current Schedule.

Show Dialog: Select this option to bring up the Schedule Dialog with the information of the current Schedule.

Schedule Page Fields

The Schedules Page shows the following information about schedules:

Name: Schedule Name

Suffix: Schedule Suffix. For a description on how the suffix is created, please see the [Schedule Dialog](#) section.

Periodic: Whether the schedule acts as periodic or not.

Period Days: Number of days in the schedule period.

Period Hours: Number of hours in the schedule period.

Period Seconds: Number of seconds in the schedule period.

Valid From: Whether a start validity date is used for the schedule.

Period Start Date: Start validity date.

Period Start Hour: Start validity time

Valid Until: Whether an end validity date is used for the schedule

Period End Date: End validity date

Period End Hour: End validity time

Schedule Subscription Dialog

This dialog is used to define and display the options of a Schedule Subscription. Schedule Subscriptions tell the Time Step Simulation that a specific object field (Gen MW, Line Status, Scheduled Transaction MW, etc) should vary according to the "shape" specified in a Schedule.

Simulator objects have a large number of fields, which can be classified as numeric, Boolean, or text. Numeric fields, whether integer or real, can subscribe to Numeric Schedules, Boolean fields can subscribe to Yes/No Schedules, and text fields can subscribe to text schedules. Thus a condition for a field to subscribe to a schedule is that a schedule of its type be already defined.

The Dialog includes the following options:

Active: Indicates that the schedule subscription is active. If this box is not checked, the schedule subscription is defined, but no schedule action will be applied to the power system.

Object Type: Use this selector to specify the object type that you want to subscribe to a schedule. Currently, Simulator supports Generators, Loads, Line/Transformers, Shunts, Areas, Zones and Transactions. When you select an **Object Type**, three things happen:

- The **Object ID** list is populated with the elements of the object type present in the power flow case and the first element is displayed by default.
- The **Field** selector is populated with the enterable fields corresponding to that particular **Object Type** and the first field is displayed by default.
- The type of the first field (Numeric, Yes/No or text) is identified and the **Schedule** selector is populated with the existing schedules of that type.

Object ID: Use this selector to specify the particular object of a particular **Object Type** whose field you want to subscribe to a schedule. The **Object ID** has above it a string that indicates how the ID of the object is built depending on the **Object Type**. For instance, when the **Object Type** is Generator, the **Object ID** is specified as the Bus Number and then the Gen ID.

Select/View Objects: Press this button to bring up the Add Schedule Subscribers Dialog. This dialog is used to select multiple objects of the specified type, for which a field will subscribe to the schedule. When multiple objects are selected, the Object ID indicates "Multiple Objects" followed by the number of objects that will be selected. If there are objects whose fields are subscribed to a schedule, these objects will appear in the right side selection list, because they are currently subscribing to the schedule.

Field: Use this selector to indicate the particular **Field** that you want to subscribe to a schedule. Depending on the type, the Dialog indicates whether the **Field** should subscribe to a Numeric, Yes/No, or text schedule. Once you have selected the **Field**, the **Schedule** selector is populated only with the schedules of that particular type.

Schedule: Use this selector to specify the schedule to which the **Field** should be subscribed to.

Time Shift: The time shift options can be used to delay applying the scheduled actions by the specified **Delay Days**, **Delay Hours**, **Delay Minutes**, or **Delay Seconds**. For instance, suppose that you have setup a schedule so that a new 100MW generating unit enters online on March 10th. The entering schedule is complex and consists of increasing the output in steps of 20MW each hour starting at 10 am. Suppose that due to unexpected events, the connection of the unit must be put off by two days. Instead of modifying the entire schedule, you could set a 2 day (positive) delay so that the unit enters online on March 12th at 10 am.

Subscription Type: In addition to the time shift, the value "shape" defined by a numeric schedule can be altered by making the subscription **Relative**.

Absolute: Select this option so that the field takes the exact value specified in the schedule.

Relative: The numeric values of the schedule can be altered by a **Multiplier** and a **Value Shift**. In this linear modification, the value of the field that is applied to the power system will be equal to:

$$\text{Actual Field Value} = \text{Multiplier} * \text{Schedule Value} + \text{Value Shift}$$

Schedule Subscriptions Page

The **Schedule Subscriptions Page** of the [Input Pages](#) is used to display all the schedule subscriptions and their properties.

Schedule Subscriptions Page Local Menu Actions

Insert New Subscription: Use this option to define a new subscription through the [Schedule Subscription Dialog](#)

Delete: This option deletes the current Subscription.

Show Dialog: This option brings up the [Schedule Subscription Dialog](#) with the information of the current Schedule Subscription.

Schedule Subscriptions Page Fields

The Schedule Subscriptions Page shows the following information:

Object: Object Type: Generator, Load, Line/Transformer, Shunt, Area, Transaction, Zone, Interface, DC Line

Object IDs: Are explained in the following Table:

Object Type	Object ID1 <i>(Numeric)</i>	Object ID2 <i>Numeric</i>	Object ID3 <i>String[2]</i>
Generator	Bus Number		Gen ID
Load	Bus Number		Load ID
Line/Transformer	From Bus Number	To Bus Number	Circuit ID
Shunt	Bus Number		Shunt ID
Area	Area Number		
Transaction	From Area Number	To Area Number	ID
Zone	Zone Number		
Interface	Interface Number		
DC Line	Rectifier Number	Inverter Number	ID

Object Field: Field that subscribes to the schedule

Schedule Name: Name part of the schedule

Suffix: Suffix part of the Schedule Name

Active: If not active, then the schedule values are not applied to the power system

Relative: If not relative the field takes the exact schedule values. If relative the schedule takes the value of the schedule scaled by a multiplied and added a Value Shift.

Multiplier: Scaling factor used when the subscription is Relative.

Value Shift: Shift value used when the subscription is Relative.

Delay/Advance?: Indicates if the schedule should be delayed or advanced to a new point indicated by the shift values.

Day Shift: Number of days of the schedule time delay

Hour Shift: Number of hours of the schedule time delay

Minute Shift: Number of minutes of the schedule time delay

Second Shift: Number of seconds of the schedule time delay

Schedule Subscriptions can be saved and loaded from auxiliary files using options found on the local menu obtained by right-clicking on the table or using the case information toolbar.

Running a Timed Simulation

The **Timed Simulation** is set up in the [Options Page Time Step Simulation Options](#) subsection.

By default, the **Do Run** button of the [Time Step Simulation Dialog](#) will run a **Continuous Simulation**, i.e., a simulation in which each time point is solved immediately after the previous time point. In this case the purpose of the Simulation is to obtain solution for the time points as fast as possible. As the solution progresses, the results for each hour in every hourly grid are refreshed, showing the user the evolution of the Simulation. Please read the [Time Step Simulation Quick Start](#) section for a quick introduction on how to run a **Continuous Simulation**.

On the other hand, the Time Step Simulation tool can also be used to run a **Timed Simulation**. In this case, the Simulation takes place according to a time scale proportional to the date times of the time points. When you click the **Do Run Button**, the Simulation progresses as if it was running in actual time. The simulation can also be paused and reset at any time by using the **Pause** and **Reset** Buttons of the [Time Step Simulation Dialog](#).

The following are some of the things you can do with the **Timed Simulation**:

- You can hide the [Time Step Simulation Dialog](#) and control the simulation by using the [Time Step Simulation Toolbar](#), which contains buttons such as **Do Run**, **Pause**, **Reset**, etc. which mimic the control buttons of the Dialog.
- You can visualize how the quantities vary proportionally to actual time after each time step is applied. Recall that the **Time Scale** defined in the [Options Page](#) indicates the relationship between the actual time and the time scale defined for the time points in Seconds per Hour. Thus a Time Scale value of 60 will indicate that one hour of time span between two time points will occur in 60 seconds of actual time in the **Timed Simulation**.
- You can also animate the flows of the solution of a time point while you wait for the solution of the next one.
- You can contour the online diagrams. And see how the visualization changes as the quantities vary in time, and in addition, you can save those contour diagrams as JPEG or bitmaps for each timepoint.
- Finally, you can act on the system (by closing capacitors, changing generator outputs, etc) before the next time point is applied simulate operating actions in response to system conditions.

All these actions can be combined in outstanding presentations to your colleagues or clients on how the system would evolve in time across different scenarios.

Running OPF and SCOPF Time Step Simulations

Besides obtaining hourly power flow solutions for multiple time points, users that own the OPF and the SCOPF add-ons can obtain hourly optimal power flow and security-constrained optimal solutions. In the Hourly Summary Page, you can specify the following solution types:

- Power Flow
- Unconstrained Optimal Power Flow, which is equivalent to Economic Dispatch
- Optimal Power Flow (OPF)
- Security-Constrained Optimal Power Flow.

Note that different timepoints can be solved by any of the previously listed solution methods in the same **Time Step Simulation**. However, the solution settings of the previous timepoints are used as initial conditions for the solution of the next time point.

Simulator OPF and SCOPF tools are among the most advanced optimization packages for power systems. They have been extended in the latest versions of Simulator with many features, and have become complex analysis system. We recommend the user unfamiliar with OPF/SCOPF solutions to read the sections on [Optimal Power Flow](#) and [Security-Constrained Optimal Power Flow](#) before setting up OPF/SCOPF Time Step Simulations.

Power Flow Time Step Simulation

The hourly power flow simulation allows the user to obtain AC or DC power flow solutions for a set of timepoints. During the Time Step Simulation all the power flow options defined in the [Simulator Options](#) page as well as in other dialogs are used for the solution. A key concept of the Time Step Simulation is that if you select **Solve Time Point** from the [Hourly Summary Page](#) or from the grids of the [Input Page](#), you would obtain the same solution that if you would hit the **Single Solution Button**. This is true, when the input data does not contain schedule data but only hourly input data. If your Simulation contains schedule data, there may have been scheduled actions that were applied in previous timepoints that are not being applied when you select the **Solve Time Point** option.

The power flow solution will observe all the power balance constraints, control limits and area interchange constrained defined in the power flow settings.

Unconstrained OPF Time Step Simulation

In the unconstrained OPF solution, the Time Step Simulation removes all the constraints that would normally act in the OPF and optimizes the system to find the minimum operating cost settings. In doing so, Simulator will change the set points of the specified controls (generators and phase shifters) to minimize the cost of all Areas and Superareas set to OPF AGC control.

Besides the power flow solution options, the Unconstrained OPF simulation will take all the options that have been defined for a regular OPF solution. Most of these options are defined in the [OPF-Options Dialog](#), which is accessed using the **OPF Options and Results** button on the **Add Ons** Ribbon tab. All the settings such as objective function, cost of unenforceable constraints, control available, prices for controls, etc are defined in this dialog. Other options are defined for each particular object such as generators, loads, areas, lines, interfaces, etc.

OPF Time Step Simulation

When using the OPF solution type, the Time Step Simulation applies the hourly and schedule input data and optimizes the control areas set to OPF to minimize cost while enforcing normal operation

constraints: transmission line thermal limits, interface limits, generator control limits, and load control limits. In doing so, the OPF algorithm detects the controls that need to be moved, the constraints that are binding at the solution point, and the unenforceable constraints, i.e., constraints that cannot be enforced with the available controls.

Some of the quantities that are of interest in the solution of the OPF algorithm are:

- **Unconstrained Generator MW Output.** Displayed in the Hourly Unconstraint Gen MW page of the [Input Page](#)
- **Final generator MW Output:** Optimal generator output at the solution points, displayed in the Hourly Final Generator MW Page
- **Change in Generator MW:** Difference between the unconstrained and constrained generator optimum settings, displayed in the Hourly Delta Gen MW Page.
- **Locational Marginal Prices:** These are displayed in the Hourly Final Bus LMP Page. Average LMP prices and other LMP metrics are also displayed in the [Results Page](#) for Areas, Injection Groups, Super Areas, and Zones.
- **Binding Constraints** as well as **Marginal Cost of Limit Enforcement** for lines and interfaces. These fields can be seen in the [Results: Constraints Page](#) and in the [Results Page](#) for Lines, Transformers, and Interfaces.
- **Unconstrained, Final and Congestion Cost**, displayed in the **Results Summary Page** for the entire system. These costs are also displayed for displayed for Areas, Owners, Superareas and Zones in the [Results Page](#).
- **LMP Profit** for Generators, Owners, and Zones are displayed in the corresponding grids of the [Results Page](#).

SCOPF Time Step Simulation

The SCOPF combines the power of Simulator's OPF with the [Contingency Analysis Tool](#) to optimize a system for minimum cost while enforcing both normal operation and contingency constraints. At each time point, the SCOPF solution provides the optimal operation of the system so that if contingencies occur they would not create security violations. The locational marginal prices created in this manner are security-constrained signals to the market.

The solution of SCOPF Time Step Simulation depends on the options that have been set up for the following tools:

- Power Flow
- Optimal Power Flow
- Contingency Analysis
- Security Constrained Optimal Power Flow
- Time Domain OPF Options

The SCOPF Time Step Simulation does the following for each timepoint:

- Applies the hourly input data to the power system
- Applies scheduled actions determined by the schedule input data.
- Solves a power flow
- If specified, solves an unconstrained optimal power flow (economic dispatch)
- Initializes the base case of the security constrained OPF by solving a power flow or an OPF
- For the initialization system conditions, solves the list of contingencies
- Solves the SCOPF optimization problem: minimizes operating cost while enforcing normal and contingent constraints.
- Displays the results in all the result grids.

The SCOPF is on its own a complex computation that often requires significant computer resources. This is due mostly to the solution of the list of contingencies and the calculation of their sensitivities. The size of the problem can be dimensioned by:

- Size of the system, given by the number of buses and the areas to be optimized. This is difficult to reduce since the optimization problem is normally defined for a certain region.
- Number of contingencies, which can be reduced by developing a contingency screening using peak loading conditions.
- Number of constraints (monitored elements), which can be reduced by selecting critical element, e.g. interfaces and higher voltage transmission lines.
- Number of timepoints in the list.

Another mechanism to speed up the computation of the PF/OPF/SCOPF Time Step Simulation is to use DC solutions in some of the internal routines:

- AC or DC power flow
- AC or DC contingency analysis. This one will produce the larger time savings.
- AC or DC SCOPF

Storing Input Data and Results

In large cases, the amount of data that can be potentially generated by a Time Step Simulation is significant since basically a full PF/OPF/SCOPF solution is available for each hour. A convenient method to store large amounts of data is to save the data in a binary file. The input data, both hourly and scheduled data, as well as the results can be stored in a **Time Series Binary File**, referred to in this help guide as the **.tsb file**.

Here is a summary of what will be stored in the **.tsb file** when you click the **Save Data Binary** button in the [Time Step Simulation Dialog](#):

- All hourly input data, defined in the [Input Pages](#) for hourly Load MW, Load Mvar, Generator MW, Generator Maximum MW, Area Total MW Load and Hourly Line Status.
- All scheduled input data, defined by the combination of Schedules and Schedule Subscriptions specified in the corresponding [Input Pages](#).
- The values displayed on the grids of the [Results: Constraints Page](#) for hourly Binding Lines, Binding Interfaces, Binding Contingencies, and the Binding Line Matrix and List.
- The customization settings defined in the pages of the [Custom Results Selection Dialog](#) for the hourly results of Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Owner, Superareas, Transformers, and Zones.
- The customized results in the pages of the [Results Page](#) for hourly field values of Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Owner, Superareas, Transformers, and Zones. Recall that many fields can be defined for each type of object.
- The options defined in the [Options Page](#), except the options set up in the **Auto Load TSB File Options**, which are saved with the **.pwb case**.
- The data of the last solution run contained in the [Hourly Summary Page](#): Skip, Processed and Solved fields.
- The **.tsb file** description

Note: Recall that the data of any Simulator grid can be copied to the clipboard and to Excel by selecting the options on the grid **Local Menu**.

Chapter 12: PowerWorld Simulator Add-On Tools

This chapter contains information on the tools available for purchase for adding additional functionality to the Simulator base package.

- Voltage Adequacy and Stability Tool (PVQV)
- Optimal Power Flow (OPF)
- Security Constrained Optimal Power Flow (SCOPF)
- Optimal Power Flow Reserves (OPFR)
- Available Transfer Capability Analysis (ATC)
- Simulator Automation Server (SIMAUTO)
- Simulator Automation Server (SIMAUTO) for Simulator version 9
- Topology Processing
- Transient Stability

Introduction to Simulator Add-On Tools

In addition to the features of the base Simulator package, various add-on tools are available. A brief introduction to each follows:

Voltage Adequacy and Stability Tool (PVQV)

The purpose of the PVQV add-on is to allow the user to analyze the voltage stability characteristics of a system. After the PVQV simulation is complete, the user can graph various system parameters. For more information, see the [PVQV Overview](#).

Optimal Power Flow Tool (OPF)

The purpose of an OPF is to minimize an [objective \(or cost\) function](#). In Simulator OPF the Linear Programming OPF algorithm (LP OPF) determines the optimal solution by iterating between solving a standard power flow and solving a linear program to change the system controls thereby removing any limit violations. For more information, see the [OPF Overview](#).

Security Constrained Optimal Power Flow Tool (SCOPF)

The OPF tool minimizes an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. The SCOPF tool takes it one step further by considering contingencies that may arise during system operation and ensuring that in addition to minimizing the objective function, no unmanageable contingency violations occur. For more information, see the [SCOPF Overview](#).

Optimal Power Flow Reserves (OPFR)

Simulator OPF Reserves is the tool used to simulate Ancillary Services Reserve Markets. For more information, see the [Optimal Power Flow Reserves Overview](#).

Available Transfer Capability Analysis Tool (ATC)

ATC analysis determines the maximum MW transfer possible between two parts of a power system without violating any limits. For more information, see the [ATC Analysis Overview](#).

PowerWorld Simulator Automation Server (SimAuto)

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on SimAuto, see the [SimAuto Overview](#).

Topology Processing

Topology Processing allows you to solve EMS-type, full-topology models in a numerically robust and transparent manner. Topology Processing extends Simulator applications traditionally used by planners so they can operate in an operations real-time environment. For more information, see the [Topology Processing Overview](#).

Transient Stability (TS)

Transient Stability analysis allows the analysis of system dynamic response to a fault on the system. For more information, see the [Transient Stability Overview](#).

PowerWorld Simulator PV/QV Overview

The PV and QV tools are only available if you have purchased the PVQV add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the PVQV version of Simulator.

PVQV, PowerWorld's voltage adequacy and stability assessment add-on, is used to analyze the voltage characteristics of a power system.

PowerWorld Corporation also offers Optimal Power Flow ([OPF](#)), Available Transfer Capability [ATC](#), Simulation Automation Server ([SimAuto](#)), and Security Constrained Optimal Power Flow ([SCOPF](#)) add-ons. For more information see .

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the base package, Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. However, with the voltage adequacy and stability (**PVQV**) add-on the user can solve multiple power flow solutions in order to generate a [PV curve](#) for a particular transfer or a [QV curve](#) at a given bus.

The PVQV functionality is accessed using the PV and QV Curves (PVQV) main menu item. The commands available from this menu are [Refine Model](#), [PV Curves](#), and [QV Curves](#).

The purpose of the PVQV add-on is monitor system voltages, or other user specified parameters, as a real power transfer is increased or reactive power is injected at selected buses. The PVQV add-on uses the Simulator built-in Newton Raphson power flow algorithm to accomplish this task. After PV and QV simulations are completed, the user can choose to graph any of the monitored system parameters, designated in [Quantities to Track](#).

PV Curves

Simulator's **PV Curves** tool provides the ability to produce plots of maximum power transfer (PV) curves for voltage at any bus in the system, as well as 2-dimensional plots of various quantities tracked during the simulation. In Run Mode, select **PV Curves...** from the **Add Ons** ribbon tab to open the [PV Curve dialog](#). The PV Curve dialog allows you to specify the quantities to be tracked during the simulation, control solution parameters, set violation boundaries, and run the PV analysis.

The PV curve process will automatically solve a sequence of power flows at incremental levels of power transfer between a source Injection Group and a sink Injection Group. [Injection Groups](#) may contain generators, loads, or a combination of both. Each **Contingency** defined in the [Contingency Analysis Tool](#) that is not flagged to be skipped will be processed and tracked as a separate **PV Scenario**, along with the pre-contingent base case, at each incremental power transfer. After each successful power flow solution, the user-specified [Quantities to Track](#) are recorded and the transfer is increased by the next increment. If a power flow solution fails, then the last successful solution for that scenario is applied and a smaller incremental transfer is applied. The process concludes after finding the maximum power transfer for the user-specified number of critical scenarios.

PV Curves Dialog

To display this dialog, go to the [Add Ons](#) ribbon tab and select **PV Curves** from the **PV and QV Curves (PVQV)** ribbon group.

The PV Curves dialog contains all of the setup and controls for processing and analyzing the PV curve analysis. The dialog is broken down into several pages:

[Setup](#)

[Quantities to track](#)

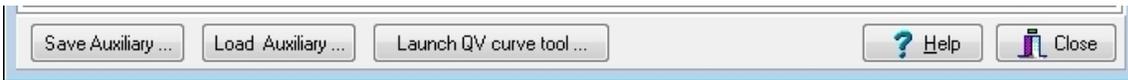
[Limit violations](#)

[PV output](#)

[QV setup](#)

[PV results](#) (Note: Launching the analysis is done from this page.)

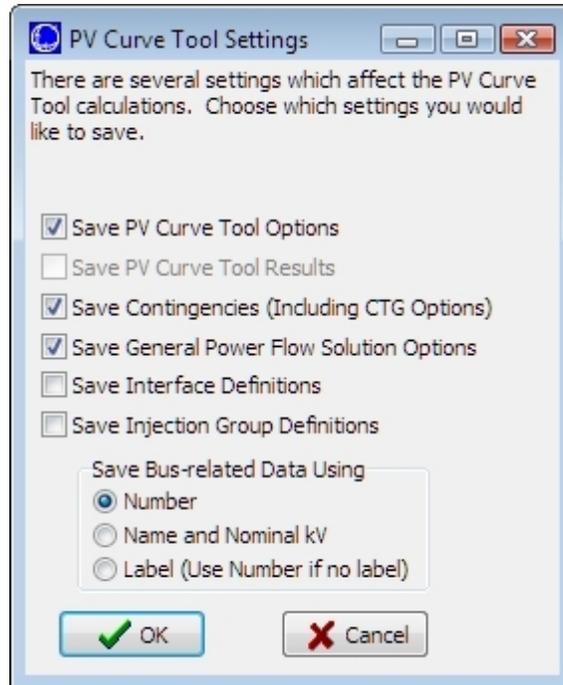
There are several buttons at the bottom of the dialog that are available regardless of the selected page.



Save Auxiliary...

Clicking this button will prompt for a filename in which to save an [auxiliary file](#). PV-specific results and option settings are not saved with the case when it is saved to file. Use this option to store any results and settings that need to be retained for future use.

The PV Curve Tool Settings dialog shown below will be displayed with options for specific data to include in the auxiliary file. Check the box next to a particular data set to save this data in the auxiliary file. Which key field to use when identifying objects in the file can also be specified on this dialog. Click **OK** on this dialog to finalize saving the file or **Cancel** to abandon the file save.

**Load Auxiliary...**

Clicking this button will open a dialog to allow the user to select an [auxiliary file](#) to load. This is intended to be used for loading any relevant option settings to be used during the PV analysis. The dialog will be updated according to the option settings loaded from the file.

Launch QV curve tool...

This button will launch the [QV Curves](#) tool using the options specified on the [QV setup](#) tab.

Help

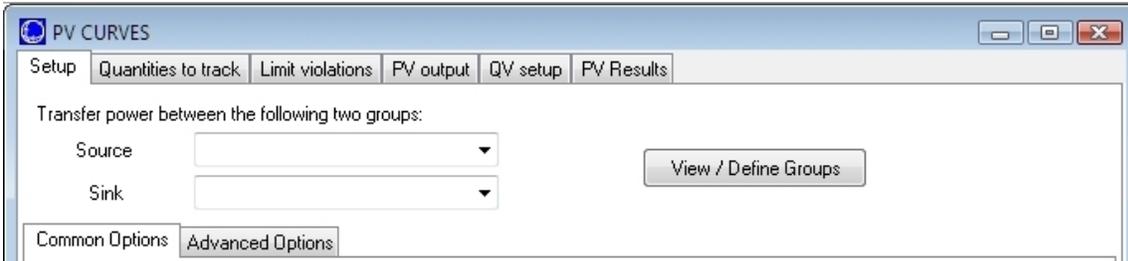
Displays this help page.

Close

This will close the dialog without running the analysis. The currently set options will be saved, and the dialog will not close unless all option settings are valid.

PV Curves: Setup

The Setup tab is found on the [PV Curves dialog](#).



The Setup tab contains the options for defining the source and sink for the PV study transfer. The PV tool expects the source and the sink to be [injection groups](#) defined by the user.

Source

Use this drop-down box to identify the source [injection group](#). To model an increase in transfer, generator points in the source injection group will increase their output, and load points will decrease their magnitude in amounts proportional to their participation factors.

If the injection group to use for the source has not already been created, click the View/Define Groups button in order to create it. After the injection group has been created, it will show up in this drop-down box and can then be selected.

Sink

Use this drop-down box to identify the sink [injection group](#). To model an increase in transfer, generator points in the sink injection group will decrease their output, and load points will increase their magnitude in amounts proportional to their participation factors.

If the injection group to use for the sink has not already been created, click the View/Define Groups button in order to create it. After the injection group has been created, it will show up in this drop-down box and can then be selected.

View/Define Groups

If injection groups have not been previously defined in the current case, they can be created by clicking on this button. To create a new injection group, right-click on the resulting injection group [case information display](#) and choose **Insert** from the popup menu.

Alternatively, if a list of injection groups has been previously saved in an [auxiliary](#) file, they can be loaded into the current case by right-clicking on the resulting injection group case information display and choosing **Load > Auxiliary File** from the popup menu.

Once the source and sink points have been defined, the PV study will model an increasing transfer of power from the source points to the sink points. The transfer process is performed incrementally, based upon user specified [options](#) on how the transfer should vary during the solution process.

The remainder of the setup options are found on sub-tabs for [Common Options](#) and [Advanced Options](#).

PV Curves Setup: Common Options

These options are located the Common Options sub-tab found on the [Setup](#) tab of the [PV Curves dialog](#). These are the options most commonly adjusted for the PV analysis.

Initial Step Size (MW)

This option indicates the initial increment in which the transfer will be increased following each successful iteration. The default value is 100 MW.

Minimum Step Size (MW)

Whenever the PV process fails to solve the system at a given transfer level, it will return to the previously solved transfer level, reduce the step size by the specified factor, and then try to solve the system with the transfer incremented by the newly reduced step size. The **Minimum Step Size** option specifies the minimum size this increment can be. Once the system fails to solve when the step size is at this value, the process will conclude that we have come very close to the voltage collapse point and terminate the analysis. So, the minimum step size essentially functions as a tolerance for computing the voltage collapse point. The default value is 10 MW.

When convergence fails, reduce step by a factor of...

Whenever the PV process fails to solve the system at a given transfer level, it will reduce the transfer step size by the value specified for this option. The default value is 2. Therefore, the process will start incrementing the transfer in 100 MW steps. When it reaches a transfer level that it cannot solve, it will return to the last solved transfer level, reduce the step size to 50 MW, increment the transfer by 50 MW, and attempt to solve the case again. The next time it fails to solve, it will reduce the step size to 25 MW, and then to 12.5 MW, and finally to 6.25 MW. Since 6.25 MW is less than the **Minimum Step Size** value of 10 MW, it will instead use a final step size of 10 MW. Once the system fails to converge with this step size, the analysis will terminate because it concludes that it has arrived at the voltage collapse point, within the specified tolerance.

Stop when transfer exceeds

Check this box to terminate the PV analysis once the transfer between the source and sink reaches an amount equal to the MW value specified.

Island-Based AGC Tolerance

When implementing the transfer, island-based AGC is used. Injection changes for the transfer are first made to the source injection group, and when the power flow is solved, the sink injection group is adjusted for any injection changes due to the source injection group changes and any changes in losses due to the transfer. Because the process to determine required injection changes during the power flow solution MW control loop is an iterative process, a tolerance value must be specified to determine when the changes to the sink injection group are acceptable. This tolerance is the same value that is set with the [power flow solution options for specifying island-based AGC](#).

As a general rule of thumb, the Minimum Step Size should be at least 1 to 2 times larger than this tolerance. If not, all changes due to an incremental transfer that is smaller than this tolerance will actually occur between the source and the system slack instead of the source and the sink.

Allow only AGC units to vary

Individual generating units are distinguished according to whether they do or do not participate in the automatic generation control (AGC) effort. Each generator has a field that determines whether or not that particular generator is AGC-able. By checking this option, only those generators whose AGC field is set to *YES* will contribute to the power transfer being studied. Otherwise, all units in the injection groups will be allowed to participate regardless of their AGC status.

Enforce unit MW limits

If this option is checked, the output of any participating generating unit will be kept within its designed operating range of $\text{MinMW} < \text{Output} < \text{MaxMW}$. When a unit is pegged at one of its limits, participation factors of the other points in the limited generator's injection group will be adjusted to pick up the difference.

Do not allow negative loads

This is the analog of the previous option for loads. If a load is used as a source point (or sink point when choosing to apply the [reverse transfer](#)), it will be decreased to make power available for the transfer. Checking this option will instruct the PV process to keep loads from falling below 0 MW. If a particular load is capped at 0 MW, participation factors for the remaining points in its injection group will be recalculated to make up the difference.

Dispatch generators in merit order

When choosing to dispatch in merit order, injection groups with generators will be dispatched by moving individual generators to their maximum/minimum outputs in succession based on their relative participation factors. To dispatch the source injection group in merit order, check the box next to **Source**. To dispatch the sink injection group in merit order, check the box next to **Sink**.

When using this option, generator unit MW limits will be enforced for those units participating in the merit order dispatch regardless of how the **Enforce unit MW limits** option is set.

Skip contingencies

The PV Curve tool computes PV curves for both the base topology and for any contingencies that have been defined, unless the **Skip contingencies** checkbox is checked. If the **Skip contingencies** checkbox is checked, a PV curve will be computed only for the model in its present (base case) topology.

Manage contingency list...

Clicking this button will open the [contingency analysis dialog](#) for managing or inserting contingencies to be processed during the PV analysis. Contingencies defined and marked for processing in the contingency analysis dialog will be included. Only those contingencies whose **Skip** field is set to *NO* will be included in the PV analysis.

Note: The Robust Solution Process option that can be utilized with the contingency analysis when a contingency solution failure occurs is not applied when a contingency is implemented during the PV analysis. All other options defined with contingency analysis such as contingency specific solution options and make-up power specifications are used when applying a contingency during the PV analysis.

Run base case to completion

The PV curve tool is designed to ramp a transfer until the prescribed number of [critical scenarios](#), including both critical contingencies and critical base topology, have been found. If the requested number of critical scenarios have all been identified as being associated with contingencies, the tool will not reveal how much a transfer can be ramped for the base topology, unless the **Run base case to completion** checkbox is checked. Checking this checkbox forces the tool to continue to ramp the transfer until the base case can no longer be solved, regardless of whether the requested number of critical transfer level / topology combinations have been found.

Base Case Solution Options ...

Click this button to bring up the [Simulator Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases and the options used when no contingency-specific solution options are defined. Changing these options affect all power flow computations, even those outside the PV process.

Because the goal is to stress the system while performing the PV analysis, there are two power flow solution options that are set internally as part of the PV process. These are the options that specify the minimum per unit voltage for constant power and constant current loads. These options are both set to 0 pu and override any user-specified settings.

Minimum Per Unit Voltage for	
Constant Power Loads	0.000
Constant Current Loads	0.000

These options are both set to 0 pu during both the base case solution and any contingency case solution.

PV Curves Setup: Advanced Options

These options are found on the Advanced Options sub-tab located on the [Setup](#) tab of the [PV Curves dialog](#).

Common Options | **Advanced Options**

How should reactive power load change during ramping?

Maintain the MW/MVAR ratio at each load, but then scale MVAR by a factor of

As MW changes, change the MVAR at a power factor of

Load Component Variation

All changes apply to constant power (S MW, S MVAR)

Vary in proportion to existing Z,I,P ratios

Vary using proportions specified below:

	P (MW)		Q (MVAR)	
	Source	Sink	Source	Sink
Power (S)	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>
Current (I)	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>
Impedance (Z)	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>

Reverse Transfer

Apply Reverse Transfer

Maximum Reverse Transfer (MW > 0)

The power transfer that occurs during the PV analysis is a real power (MW) transfer. Loads can be specified as part of either the source or sink injection group that define the transfer. When loads are included in the transfer, the default is to only adjust the real power (MW) component of the load and leave the reactive power (Mvar) component constant. If any change is made to the reactive power load (Mvar) during the transfer, the default is to have all changes be made to the constant power component of the load. The Advanced Options tab allows specification of load variation for both real and reactive load to be something other than the defaults during the transfer.

The transfer amount and how the transfer is adjusted are determined by options found on the [Common Options](#) tab.

How should reactive power load change as real power load is ramped?

Total real power load changes, ΔP , at each load are determined based on the load's participation factor as defined with its associated injection group and the amount of transfer that is being implemented. ΔP is then used in conjunction with the selected reactive power change option to determine the associated total reactive power change, ΔQ , at each load.

Maintain the MW/MVAR ratio at each load, but then scale MVAR by a factor of

The total reactive power change, ΔQ , is determined based on the power factor, pf_i , at each nominal load (P_{nom}, Q_{nom}) prior to the load change and the change in total real power load, ΔP , due to the transfer. The power factor is determined by the following:

$$pf = \cos \left(\tan^{-1} \left(\frac{Q_{nom}}{P_{nom}} \right) \right)$$

An additional multiplier, $pfQMult$, can be specified in order to modify the total reactive power load change to allow adjustment away from the present power factor. This multiplier is 1.0 by default. The change in reactive power load is then:

$$\Delta Q = \tan\left(\cos^{-1}(pf)\right) * \Delta P * pfQMult$$

Maintaining a constant MW/MVAR ratio implies a sign convention that is lost when applying the arctan function to the ratio. An additional check is done to make sure that the calculated ΔQ is in the correct direction (maintains the correct leading or lagging power factor).

$$\text{if } \frac{Q_{nom}}{P_{nom}} < 0 \text{ then } \Delta Q = -1 * \Delta Q$$

$$\text{if } P_{nom} = 0 \text{ then } \Delta Q = 0$$

When loads are specified as constant power, constant current, and constant impedance components (ZIP components) rather than simply all constant power, the total load becomes a function of voltage. Maintaining a constant power factor at each load requires that we use nominal load (load at 1.0 pu voltage) that does not change as a function of voltage. When only constant power components are specified, the actual load is the same as the nominal load. Therefore, in all situations the power factor is determined based on the nominal load.

As MW changes, change the MVAR at a power factor of

This option allows the specification of power factor, $pf_{specified}$, for defining the total reactive power change, ΔQ , at a load. The change in reactive power load is then dependent on the change in total real power load due to the transfer, ΔP , and this specified power factor.

$$\Delta Q = \tan\left(\cos^{-1}(pf_{specified})\right) * \Delta P$$

For this option, ΔQ is assumed to be in the same direction (has the same sign) as ΔP .

Load Component Variation

The total real power load change, ΔP , at a load during a PV transfer is determined by the load's participation factor as defined with its associated injection group and the amount of transfer that is being implemented. The total reactive power load change, ΔQ , is determined by the option selected for how reactive power should change during ramping as explained in the previous section. Total load change, $(\Delta P, \Delta Q)$, at a load can be broken into ZIP components for constant power $(\Delta P_s, \Delta Q_s)$, constant current $(\Delta P_I, \Delta Q_I)$, and constant impedance $(\Delta P_Z, \Delta Q_Z)$. During the PV transfer, the user can specify how changes in load due to the transfer should be split among these components.

The constant current and constant impedance components are a function of voltage and the nominal values specified for these components. Because of this, it becomes important to determine the load changes for the components based on nominal voltage and then apply these changes to the nominal components.

All changes apply to constant power (S MW, S MVAR)

All changes to load will be made to the constant power component:

$$\Delta P_s = \Delta P, \Delta P_I = 0, \Delta P_Z = 0, \Delta Q_s = \Delta Q, \Delta Q_I = 0, \Delta Q_Z = 0$$

The constant power component of load is not dependent on voltage so the total load change can be applied directly to the constant power component.

Vary in proportion to existing Z,I,P ratios

This option will change the load at a given load such that the ratios of the ZIP components do not change during the load adjustment. The existing ZIP ratios at a load are determined based on the present nominal load prior to any change due to the transfer. The existing total nominal power (P_{nom}, Q_{nom}) at a load is the sum of the components:

$$P_{nom} = P_{nomS} + P_{nomI} + P_{nomZ}$$

$$Q_{nom} = Q_{nomS} + Q_{nomI} + Q_{nomZ}$$

The component ratios are simply the ratio of each nominal component to the total nominal load at a load:

$$k_{PS} = \frac{P_{nomS}}{P_{nom}}, k_{PI} = \frac{P_{nomI}}{P_{nom}}, k_{PZ} = \frac{P_{nomZ}}{P_{nom}} \quad \text{if } P_{nom} \neq 0$$

$$k_{QS} = \frac{Q_{nomS}}{Q_{nom}}, k_{QI} = \frac{Q_{nomI}}{Q_{nom}}, k_{QZ} = \frac{Q_{nomZ}}{Q_{nom}} \quad \text{if } Q_{nom} \neq 0$$

$$k_{PS} = 1, k_{PI} = 0, k_{PZ} = 0 \quad \text{if } P_{nom} = 0$$

$$k_{QS} = 1, k_{QI} = 0, k_{QZ} = 0 \quad \text{if } Q_{nom} = 0$$

The resulting change in nominal load for each load component at a load is then the product of the ratio of each component and the total nominal power change:

$$\Delta P_{nomS} = k_{PS} * \Delta P_{nom}$$

$$\Delta P_{nomI} = k_{PI} * \Delta P_{nom}$$

$$\Delta P_{nomZ} = k_{PZ} * \Delta P_{nom}$$

$$\Delta Q_{nomS} = k_{QS} * \Delta Q_{nom}$$

$$\Delta Q_{nomI} = k_{QI} * \Delta Q_{nom}$$

$$\Delta Q_{nomZ} = k_{QZ} * \Delta Q_{nom}$$

Where the total nominal power change is determined from the total power change required at a load as a function of the present voltage, V , at the bus:

$$\Delta P_{nom} = \frac{\Delta P}{k_{PS} + k_{PI} * V + k_{PZ} * V^2}$$

ΔQ_{nom} is calculated from ΔP_{nom} based on the selection of the option on how reactive power should change during the ramping as described in the previous section.

Vary using proportions specified below:

The factors are divided into four groups: real power load in the source injection group, real power load in the sink injection group, reactive power load in the source injection group, and reactive power load in the sink injection group. Each of these groups determines how the various components will change based on whether the load change is being done to real or reactive load and whether the load is part of the source or sink injection group. The default setting for each group is to apply changes only to the constant power component; **Power (S)** factor is 1 while all other factors are 0. The sum of the factors for each grouping must add up to 1. The

Impedance (Z) factors are not enterable and are calculated to ensure that the sum is always 1. Simply adjust the **Power (S)** and **Current (I)** factors to appropriate values and the **Impedance (Z)** factor will be set automatically.

The calculation for determining the change in load components is done the same as described above when using existing ZIP ratios except that the ratios are user-specified here.

Reverse Transfer

The standard way of performing a PV analysis is to increase a transfer from the source to the sink in positive step increments determined by the Initial Step Size and other parameters that control the size of the step. When a contingency does not solve at zero transfer (base case conditions), it might be possible to find a solvable point by adjusting generation and/or load in the system. The Reverse Transfer option will attempt to find this solvable point by increasing a transfer from the sink to the source (instead of source to sink). This is done by incrementing the transfer from the source to the sink in negative step increments, effectively making the transfer go from the sink to the source.

When the **Apply Reverse Transfer** box is checked, an attempt will be made at finding a solvable point by applying the reverse transfer (sink to source) for any contingency that will not solve at zero transfer. A condition for this option to be used is that the base case power flow must solve.

When choosing to attempt the reverse transfer for contingencies that will not solve, all unsolvable contingencies will be processed even if the number of unsolvable contingencies exceeds the number of critical scenarios specified on the [Results](#) tab. The reverse transfer process will be done for all unsolvable contingencies prior to the standard PV process. After the reverse transfer process completes, the standard PV process (positive transfer from source to sink) will be performed on all contingencies that do solve at zero transfer level as long as the number of critical scenarios specified is not exceeded already by the number of contingencies that wouldn't solve as zero transfer.

The initial step magnitude by which the transfer is increased is the Initial Step Size specified with the Common Options. When a transfer level is found at which a contingency will solve, the step magnitude is reduced by the reduction factor specified with the Common Options. This is done in an attempt to find the minimum transfer required to reach a solvable point. This process will continue until a solvable point is found for each contingency of the **Maximum Reverse Transfer (MW > 0)** threshold is exceeded. It is likely that no solvable point can be found for some contingencies. The Maximum Reverse Transfer threshold should be set such that a reasonable transfer amount is attempted before abandoning the attempt at finding a solvable point. This process could be very time consuming if there are a number of contingencies that do not solve at zero transfer.

In addition to attempting to find a solvable point, the reverse transfer process can also attempt to find a solvable point at which all voltages are considered adequate. To check for adequate voltages during the reverse transfer process, check the option to **Stop when voltage becomes inadequate** that is found on the [Limit Violations](#) tab of the PV Curves dialog. This will force the process to find a reverse transfer point at which the contingency solves and all voltages are above the voltage level specified as inadequate. The inadequate voltage level is specified with the **Voltage level to consider inadequate** option also found on the Limit Violations tab.

If a reverse transfer point is found at which a contingency will solve, and voltages are adequate if choosing to include that check, the [results](#) for the scenario will list the **Critical Reason** as *SRT - original critical reason* with *original critical reason* being the reason why the contingency was considered not to be solved at the zero transfer level. The **Max Export**, **Max Import**, and **Max Shift** values that are reported are the minimum (in magnitude) transfer levels at which the contingency will solve and any voltage conditions are met. Values for any parameters that are being tracked through [Quantities to Track](#) will also be recorded at this transfer level. If there are multiple steps at which the contingency will solve and any voltage conditions are met while trying to find the minimum step, multiple transfer level points will be recorded making it possible to plot a few points of a PV curve.

If during the reverse transfer process no point can be found for which a contingency will solve and voltage conditions are met, the original results will be reported as if the reverse transfer process was not attempted.

If enforcing generator MW limits and/or not allowing negative loads during the transfer causes either the source or sink to not have enough resources to meet the required transfer, two other Critical Reason messages are possible. If the sink is maxed during the transfer, the **Critical Reason** will be given as *RT Sink Maxed - original critical reason*. If the source is maxed during the transfer, the Critical Reason will be given as *RT Source Maxed - original critical reason*. The **Max Export**, **Max Import**, and **Max Shift** values reported will be the values at which either the source or sink hit limits. Either of these messages indicates that the contingency will not solve or voltage conditions are not met, but the reverse transfer cannot continue because there are not enough resources in either the source or sink to do so. If either the source or sink is at its limits the contingency solution will still be attempted. If the contingency does solve and the voltage conditions are met, the **Critical Reason** will be given as *SRT - original critical reason*.

PV/QV Quantities to Track

Specification of quantities to track during analysis is almost identical for both the PV and QV analysis (exceptions will be noted below). For PV analysis, the Quantities to Track tab is accessed from the [PV Curves](#) dialog, and for the QV analysis, this tab is accessed from the [QV Curves](#) dialog. Quantities to track are monitored and stored as the transfer increases for the PV analysis or the voltage set-point changes for the QV analysis. Any unselected system parameters will not be saved. The Quantities to Track page contains several sub-tabs that allow the monitoring of different types of objects including: buses, generators, injection groups, lines, transformers, shunts and interfaces. There is also a sub-tab for monitoring devices at limits.

Buses

Voltage - per unit voltage

kV Voltage - actual voltage in kV

Angle - voltage angle in degrees

MW Load - total real power load

Mvar Load - total reactive power load

dV/dQ - sensitivity of voltage to change in reactive power injection at the same bus

VP Sensitivity - sensitivity of voltage change due to real power transfer. For PV analysis this is the sensitivity of the voltage at the bus due to the selected transfer from source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Shunt Inj - total switched shunt reactive power (Mvar) output

The default value of all bus quantities is *NO*, indicating that the analysis will not monitor any bus-related quantities. To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

For PV analysis if the intent is to trace PV curves, it makes sense to monitor at least one bus voltage. The process will not automatically monitor and store voltage quantities.

For QV analysis, per unit voltage will be monitored automatically for all buses selected to be included in the QV analysis.

Generators

Gen MW - real power output

Gen Mvar - reactive power output

Mvar Reserve - reactive power reserves in the positive direction (Max Mvar - Mvar)

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

Groups

This allows the monitoring of injection group parameters.

Mvar Reserve - total reactive power reserves in the positive direction for all generators in the group (Max Mvar - Mvar)

Gen MW - total real power output of all generators in the group

Gen Mvar - total reactive power output of all generators in the group

Load MW - total real power load for all loads in the group

Load Mvar - total reactive power load for all loads in the group

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

Lines

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab.

Branch MW - real power flow on the branch in either the FROM-TO or TO-FROM direction

Branch Mvar - reactive power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MVA - total power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MW Loss - real power loss

Branch Mvar Loss - reactive power loss

Branch PTDF - [Power Transfer Distribution Factor](#) on the branch due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Xfmr Tap - tap ratio if the branch is a transformer

For the flow (Branch MW, Branch Mvar, and Branch MVA) and Branch PTDF fields, double-clicking on a particular entry will toggle its value from *NO* to *FROM-TO*, and double-clicking again will toggle its value to *TO-FROM*. Double-clicking on a loss field or Xfmr Tap field will toggle the value between *YES* and *NO*.

Transformers

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab. See **Lines** above for setting up common branch quantities. Only branches containing transformers will appear on the **Transformers** sub-tab.

Type - set the transformer type (Fixed, LTC, Mvar, or Phase)

Reg Val - regulated value for the transformer control

Tap Pos - tap position

Reg Err - deviation of the regulated value from the regulation Min/Max range

All fields specified above except for the Type field are used to determine which quantities should be monitored and stored during the analysis. The Type field can be accessed here in order to change the type of transformer. The type will not be monitored during the analysis.

Shunts

This allows the monitoring of switched shunt parameters.

Actual Q - actual reactive power output

Actual P - actual real power output

Nom Q - nominal reactive power output

Nom P - nominal real power output

Reg Err - deviation of the regulated value from the regulation Min/Max range. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

Reg Val - value of regulated quantity at the regulated bus. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

Interfaces

MW Flow - real power flow

Mvar Flow - reactive power flow

MVA Flow - total power flow

MW Loss - real power losses

Mvar Loss - reactive power losses

Interface PTDF - [Power Transfer Distribution Factor](#) on the interface due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

ATC Extra Monitors

This option is only available when using the PV Curves tool. The following will appear at the bottom of the Quantities to Track tab accessed from the PV Curves dialog.

Include ATC Extra Monitors

Define ATC Extra Monitors

[ATC Extra Monitors](#) are typically used with the [ATC add-on tool](#) to monitor special branches or interfaces in addition to ones monitored by setting the [Limit Monitoring Settings](#). This would be done for example if trying to determine an interface flow limit so that interface can be used as a proxy for limiting the transfer between a source and a sink. This same functionality is desirable when using the PV analysis. Both the ATC and PV analyses are similar in that they study the impacts of real power transfers. The ATC analysis is concerned with thermal limitations while the PV analysis looks at voltage stability limitations. Use of the ATC Extra Monitors with the PV analysis could be additionally useful for developing a proxy interface to help prevent voltage stability problems.

Click the **Define ATC Extra Monitors** button to view a list of existing ATC Extra Monitors or create new ones. To include monitoring of these elements at part of the PV analysis, check the **Include ATC Extra Monitors** box.

When monitoring ATC Extra Monitors with the PV analysis, the real power (MW) flow on the specified branches and/or interfaces will be monitored at each base case transfer level. No monitoring is done with contingencies implemented.

ATC Extra Monitors can be added to the Overview table, available on the [Results](#) tab, via the [Display/Column Options...](#) dialog. When displayed in this table, the value reported with the extra monitor is the flow on the extra monitor at the Max Shift level of transfer for the given scenario. Values for ATC Extra Monitors will only be displayed in the Overview table for critical scenarios.

Access to the ATC Extra Monitor values stored at transfer levels other than the Max Shift transfer for critical scenarios and values stored for non-critical scenarios is available through SUBDATA sections of the PWPVResultListContainer data type that is saved when saving PV results to an auxiliary file or in the output file saved when choosing to save results to file from the [PV output](#) tab.

Devices at Limits

The **Devices at Limits** sub-tab allows the selection of various options to track devices that hit or back off limits during the PV analysis. Generators and switched shunts at var limits, LTC transformers at tap limits, and lines and interfaces at thermal limits can all be tracked.

Track Hitting and Backing Off Limits during Ramping

<input type="checkbox"/> Generators Var Limits	<input type="text"/>	▼	<input type="button" value="Define..."/>
<input type="checkbox"/> Switched Shunts Var Limits	<input type="text"/>	▼	<input type="button" value="Define..."/>
<input type="checkbox"/> LTC Transformers Tap Limits	<input type="text"/>	▼	<input type="button" value="Define..."/>
<input type="checkbox"/> Lines Thermal Limits	<input type="text"/>	▼	<input type="button" value="Define..."/>
<input type="checkbox"/> Interfaces Thermal Limits	<input type="text"/>	▼	<input type="button" value="Define..."/>

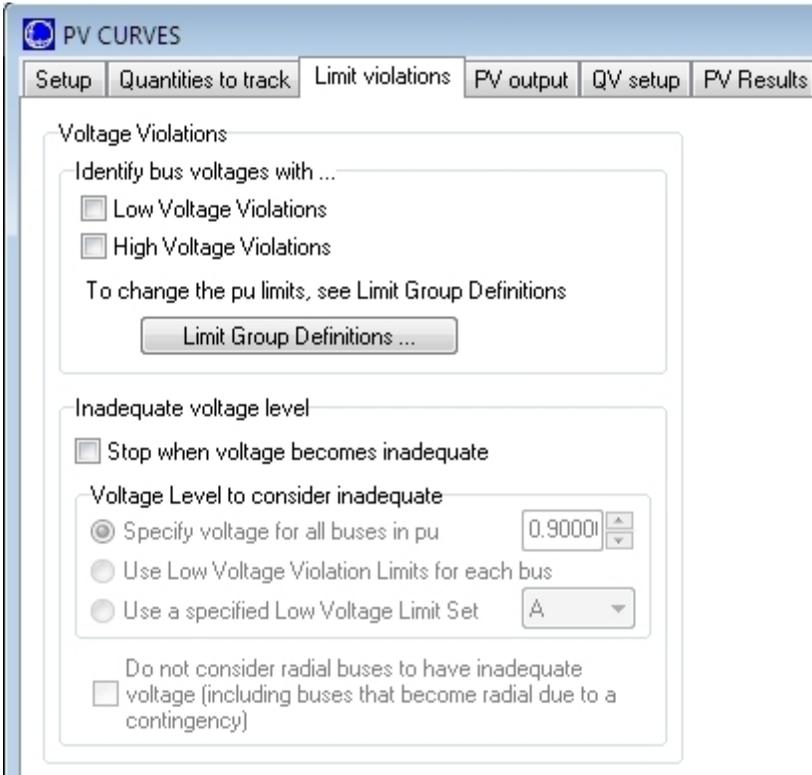
Note: Narrow limit tracking by specifying a filter
 Note: If no filter is specified, then all objects will be tracked.

To track the limits of any of these devices, check the appropriate checkbox. Next to each checkbox there is a dropdown box for selecting an [advanced filter](#) to apply to the tracking. A new filter can be defined for a particular device by clicking the **Define** button next to that device. The filter will limit the number of devices that get tracked. It is a good idea to define a filter for tracking devices so that all elements in the case will not be tracked. Keep in mind that tracking any quantities will take up space in the computer memory. If too many devices are tracked, it is possible to run out of memory.

With these options, device limits are only tracked in the base case at valid transfer levels. No device limit tracking is done with contingencies implemented.

PV Curves: Limit Violations

The Limit violations tab is found on the [PV Curves dialog](#) and allows the definition of what should be considered a violation of a monitored voltage. It also provides specification of parameters that will force the PV process to stop if voltages become inadequate.



Identify bus voltages with...

These options will store monitored bus voltages that violate either their high or low voltage limit as of the last successful solution for each scenario. Only those buses that are set to be monitored based on the [Limit Monitoring Settings](#) will be monitored, and the low and high voltage limits for each bus are also determined by the limit monitoring settings.

Low Voltage Violations - check this box to store the buses that have low voltage violations

High Voltage Violations - check this box to store the buses that have high voltage violations

Limit Group Definitions - click this button to open the [Limiting Monitoring Settings dialog](#)

When choosing to identify limit violations, the Overview table on the [Results](#) tab will contain information about the number of violations (#Viol), worst voltage violation (Worst V Viol), and bus at which the worst voltage violation occurs (Worst V Bus). Also, for any scenario in which a violation occurs, the **Show Violations** option is enabled on the local right-click menu of the Overview table. Choosing this option will open a table detailing all of the violations.

Inadequate voltage level

Stop when voltage becomes inadequate

Checking this box will stop the PV process for a scenario when any monitored bus voltage falls below the inadequate voltage threshold. The [Limit Monitoring Settings](#) dictate which buses are monitored. The inadequate voltage threshold is determined by the following options:

Specify voltage for all buses in pu - specify a single inadequate voltage threshold to be used for all monitored buses

Use Low Voltage Violation Limits for each bus - use the limit monitoring settings

Use a specified Low Voltage Limit Set - select a limit set to specify the limit from the drop-down. Up to four (A-D) specific limit sets can be specified for a bus. When using this option, all monitored buses should have the selected limit set specified. If not, the default low voltage limit will be -1.

Do not consider radial buses to have inadequate voltage (including buses that become radial due to a contingency) - check this box to exclude monitoring buses that are radial regardless of whether they are radial because they are connected to the system by a single line or because they are radial because they are connected to the system via a single in-service line (i.e. they could have more connections but all but one of the connections is open).

This option works somewhat in conjunction with a similar option found on the [Limit Monitoring Settings dialog](#). The option **Do not monitor radial lines and buses** found on the Limit Monitoring Settings dialog will exclude any bus from monitoring if it is radial in the base case. This limit monitoring settings option considers buses to be radial if they are connected by a single branch and only a single branch. When the limit monitoring settings option is selected, radial buses by that definition will be excluded from the PV monitoring. When the PV option is selected, buses that are connected by only a single in-service branch after the application of any scenario contingency, or in the base case if there is no contingency, are considered radial and will be excluded from the monitoring. The PV option is enforced regardless of whether or not the option with the limit monitoring settings is selected.

PV Curves: Output

The Output tab is found on the [PV Curves dialog](#) and allows the designation of where data should be logged.

During the PV analysis, the value of each monitored quantity is recorded at each solved transfer level. This data will only be present in memory unless some output file is specified.

Specifying File for Results

Save results to file

Click this option to save results to a file. In the adjacent text box supply the complete path and name for the output file. Click the **Browse** button to open a save dialog that will allow selection of the output file. Click the **View** button to open the specified output file in a text editor. The **View** button will be enabled if the specified output file exists.

The results can either be saved as a text file with .TXT extension or a comma-separated variable file with .CSV extension. If using the save dialog opened with the Browse button, the Save as type field on the save dialog should be set to reflect the desired file type.

When saving results to a file, a log file will also be created that indicates how long the entire PV process took and the different base case and contingency scenarios examined at each nominal shift level. This log file will use the same file name specified for the output file with "_LOG" appended to the end of the file name. The log file will be saved with .TXT extension. If running multiple instances of Simulator for PV analysis, a unique file name must be given to each set of results to avoid overwriting results created from other instances.

Single Header File

Two file formats are available when saving results to file. If using the default output format (Single Header file option unchecked), each scenario will be written as a section with a header line at the beginning of each section indicating what fields are included. Scenarios that become critical during the analysis will be listed with the base case scenario at the shift level at which they became critical. An example of this format is given below:

Example for Default PV Output (CSV):

```
RESULTS FOR PV STUDY "PVSTUDY" *
*Scenario** "base case"
"Export", "Import", "Bus 3 (Three_138.0) PU Volt", "Bus 5 (Five_138.0) PU Volt", "Newly Critical"
0.0000, 0.0000, 0.9927, 1.0066,
100.0000, -98.7365, 0.9928, 1.0062,
200.0000, -193.4265, 0.9928, 1.0054,
300.0000, -278.6270, 0.9928, 1.0043,
400.0000, -365.9341, 0.9927, 1.0031,
500.0000, -449.7520, 0.9926, 1.0015,
599.9999, -525.2490, 0.9924, 0.9996,
650.0000, -563.8931, 0.9923, 0.9986,
674.9999, -582.8196, 0.9923, 0.9980, L_00002Two-00006SixC1 (Reached Nose)
```

If choosing to use a **Single Header File** by checking this option, only a single field header will be shown at the top of the file. The first line of the file indicates that these are the results for a

PV study. Following the header are lines indicating scenarios that have become critical during the analysis. Following this information is the single header line that indicates what fields are provided with the results. Following the header are the results for each scenario. An example is given below:

Example Single Header File Output (CSV):

```
RESULTS FOR PV STUDY "PVSTUDY"
"Newly Critical","L_00002Two-00006SixC1 (Reached Nose)"
"Scenario","Export","Import","Bus 3 (Three_138.0) PU Volt","Bus 5 (Five_138.0) PU Volt"
base case, 0.0000, 0.0000, 0.9927, 1.0066
base case, 100.0000, -98.7366, 0.9928, 1.0062
base case, 200.0000, -193.4267, 0.9928, 1.0054
base case, 300.0000, -278.6272, 0.9928, 1.0043
base case, 400.0000, -365.9344, 0.9927, 1.0031
base case, 500.0000, -449.7524, 0.9926, 1.0015
base case, 599.9999, -525.2494, 0.9924, 0.9996
base case, 650.0000, -563.8935, 0.9923, 0.9986
base case, 674.9999, -582.8201, 0.9923, 0.9980
L_00001One-00002TwoC1, 0.0000, 0.0000, 0.9906, 1.0067
L_00001One-00002TwoC1, 100.0000, -98.7366, 0.9882, 1.0062
L_00001One-00002TwoC1, 200.0000, -193.4267, 0.9849, 1.0053
```

State Archiving

In addition to saving how the monitored quantities vary with the transfer, the entire system state can be saved to file at regular intervals during the analysis. This can be helpful if for analyzing particular transfer levels more closely after the analysis is complete. Keep in mind that, depending on the size of your system, archiving states frequently can require significant disk space and will delay the process. The following options are available for archiving states:

Do not save system states

This is the default option. No system states will be saved to file.

Save only the base case for each critical contingency

The base case state without the contingency implemented will be saved to file for each scenario that is critical. The state will be saved at the transfer level that is reported as the critical Max Shift value as given in the [Results](#).

Save all states

Choosing this option will save to file all scenarios at each valid transfer level. Contingency scenarios will be saved with the contingency implemented, unless at the critical transfer level. If a scenario is critical, the base case state without the contingency implemented will be saved at the critical transfer level. For critical scenarios, the critical transfer level is the value that is reported as the Max Shift value given in the [Results](#).

Save state as

Use this drop-down to select the file type that should be used when archiving states. Files can be saved as [PowerWorld auxiliary files \(*.aux\)](#), [PowerWorld binary files \(*.pwb\)](#), or both.

Where should they be stored?

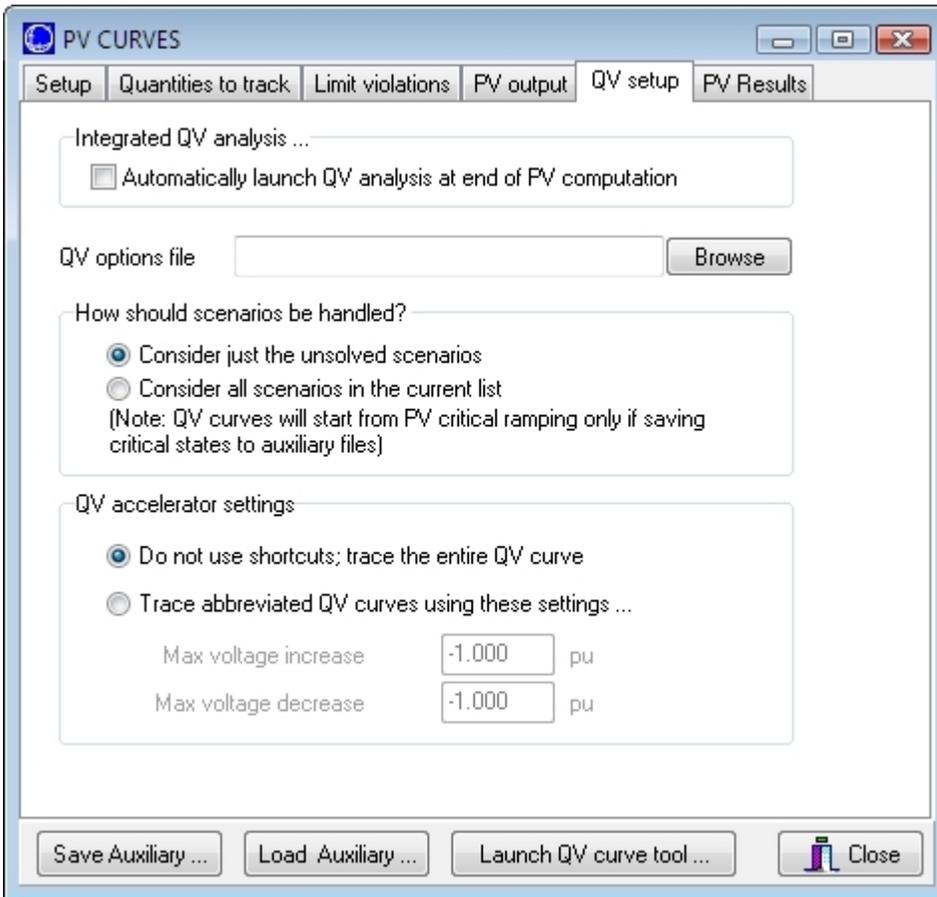
Specify the directory where state files should be saved. Click the **Browse** button to open a dialog to use for selecting the directory.

Specify a prefix to use in naming the state archives

Saved state files will be distinguished by a naming convention based on the scenario name and transfer level or critical state indicator and scenario name. Use a prefix to further distinguish these files if saving multiple study results to the same directory.

PV Curves: QV Setup

The QV Setup tab is found on the [PV Curves dialog](#). This page provides options for setting up a [QV analysis](#) and for automatically launching the QV analysis upon completion of the PV analysis.



The PV curve tool stresses the system by increasing a transfer between injection groups. Eventually, the transfer is increased so much that the power flow can no longer be solved. Under the assumptions of static voltage stability assessment, the point at which the system becomes unsolvable is regarded as the point of voltage collapse. Voltage collapse tends to be a localized phenomenon associated with a lack of reactive voltage support at a bus or group of connected buses. It is important to identify this group of stressed buses so that efforts can be properly directed to reinforce the system. QV analysis is often used to identify the buses that are most prone to a voltage collapse because they are close to having insufficient reactive support.

The design of the PVQV add-on to Simulator acknowledges the important role that QV analysis plays in identifying the buses that are most heavily stressed by a transfer. The approach is straightforward. First, perform a PV curve analysis that models a transfer from source to sink for both base and contingency topologies. This will yield a number of PV curves that terminate at a transfer level that causes the system to become unsolvable for each topology. For each of these critical transfer level / contingency pairs, perform a QV analysis at a set of buses to try to determine which area or areas constitute the epicenter of the collapse. The set of buses at which the QV analysis is performed can be

predefined by the user and supplemented by the list of buses that have the lowest voltage magnitudes or the highest VQ sensitivities.

To specify that the QV analysis automatically be launched after finishing a PV study, check the box labeled **Automatically launch QV analysis at end of PV computation**. After the PV curve has found all the critical cases it has been asked to identify, it will launch the [QV Curve Tool](#). The buses the QV Curve Tool will analyze will then come from two groups: those that the user has pre-selected, and those that tool automatically identified because they are among the lowest-voltage buses or have the highest VQ sensitivities. Whether or not to include the lowest-voltage buses or highest VQ sensitivity buses are options that need to be set by the user. Pre-selected buses for the QV analysis must also be set by the user. All of these options need to be specified in the **QV options file** that is required to be specified in order to automatically launch the QV analysis.

The remainder of the options that must be specified in order to perform the QV analysis automatically are described below:

QV options file

Specify the name of the file that contains settings for a variety of options associated with the QV study in this text box. Use the **Browse** button to locate this file using a file dialog. The QV options file should specify which buses should be selected for which to draw QV curves.

Among the topics that may be addressed by the QV options file is whether buses should be automatically selected as buses for which to draw QV curves based on their voltage or VQ sensitivity. The easiest way to build a QV options file is to use the [QV Curve Tool](#) to specify the settings of the various options and to save these settings in a file.

How should scenarios be handled?

QV curves can be drawn for the selected buses either for all the scenarios that the PV tool has processed (select **Consider all scenarios in the current list**) or just for the scenarios that the PV Curve tool determined to be critical (select **Consider just the unsolved scenarios**).

The transfer level at which a scenario is studied with the QV curve tool is based on other options that can be set with the PV tool. If choosing to archive states, and at a minimum archiving the critical scenarios, as set on the [PV Output](#) tab, critical scenarios will be studied at the last transfer level at which each scenario was successfully solved. Another condition is that the states must be saved as PowerWorld auxiliary (*.aux) files.

If not choosing to archive critical states, the critical state auxiliary file does not exist, or studying non-critical scenarios, the transfer level at which a scenario is studied in the QV curve tool is the last transfer level that the PV tool attempted to model. This transfer level is the level that is in place in the state that the case is left in upon completion of the PV analysis.

QV accelerator settings

This group of controls allows the definition of a voltage window over which each QV curve should be calculated.

Do not use shortcuts; trace the entire QV curve

Each QV curve will be computed over the voltage range specified for the bus in the **QV options file**. This voltage range may be specific to that bus or may be a window what has been defined for the entire run.

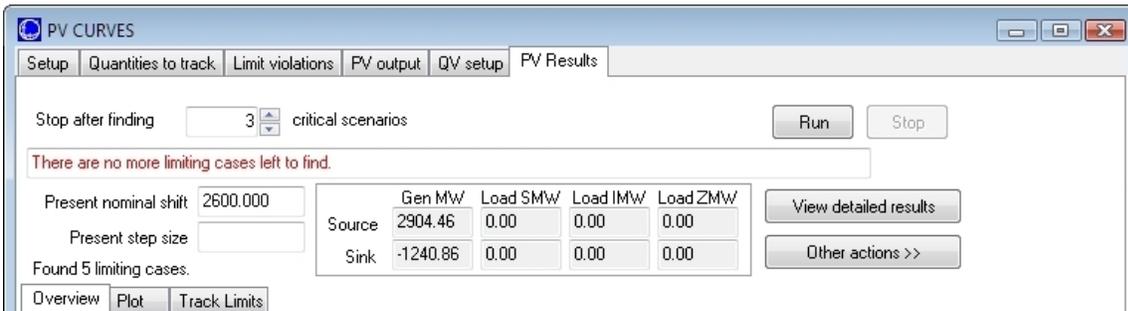
Trace abbreviated QV curves using these settings...

The voltage range over which each bus's QV curve will be computed will be customized according to the values specified for **Max voltage increase** and **Max voltage decrease**. For example, if a bus is currently at 0.9 pu voltage for a particular scenario and the max voltage increase is

specified to be 0.1 with the max voltage decrease specified to be 0.3, the QV curve for that bus for that scenario will be traced over the voltage range 0.6 pu to 1.0 pu.

PV Curves: PV Results

The PV Results tab is found on the [PV Curves dialog](#). This is the page from which the PV analysis is launched and provides access to all of the results. This page is divided into some general information at the top and sub-tabs at the bottom that provide details about the results.



Stop after finding ... critical scenarios

A large number of contingencies can be specified for analysis. Usually, there is only concern with determining a certain number of the most critical scenarios instead of running each scenario to its critical point. Set this number to stop the analysis of all scenarios once the specified number of critical scenarios have been found. It should be noted that this number really serves to find *at least* the specified number of critical scenarios. Depending on when a scenario becomes critical in the process, it is possible that more than the specified number of critical scenarios will be found.

Run

Click this button to start the analysis.

Stop

Click this button to stop the analysis once it has been started. This button will only be enabled if the analysis has been started.

Status information

Status messages will be displayed as scenarios are examined and the power transfer changes.

Summary information about the transfer

The **Present nominal shift** and **Present step size** fields will provide information about the present power transfer being examined. **Gen MW** and **Load MW** totals for both the source and sink injection group will also update as the power transfer changes to give an indication of the status of each group.

View detailed results

Click this button to open a text editor that contains detailed solution results. The results are in the same format as the output file described on the [Output](#) tab.

Other actions >>

Clicking this button opens a menu that provides access to a number of other options:

View activity log

This option will be enabled if choosing to save the output to a file via options found on the [Output](#) tab. If enabled and selected, the log file will be displayed.

View detailed results

This option does the same thing as the **View detailed results** button described above.

Clear results

This option will destroy all of the results. Nothing is done to the system state.

Save critical contingencies

Only those contingencies for critical scenarios will be saved to an auxiliary file. Selecting this option will open a dialog prompting for a file name. In addition to contingency definitions, additional data including limit monitoring settings, contingency solution options, and power flow solution options will be saved.

Restore initial state

When the PV analysis first starts running, the present system state is stored in memory and identified as the initial state. When the analysis completes, the state that the case is left in is the last analyzed scenario with the transfer in place but any contingency removed (i.e. base case state at the critical transfer level). Choose this option to return back to the initial state.

The initial state can also be set by the **Set current state as initial** option. If this is done, choosing to restore initial state will return back to whatever the set initial state was.

Restore last solved state

When the analysis completes, the state that the case is left in is the last analyzed scenario with the transfer in place but any contingency removed (i.e. base case state at the critical transfer level). This option can be used to set the state to the last solved state. The last solved state is the base case state at the last transfer level at which it was studied and solved. This transfer level will most likely be higher than the transfer level at the last studied critical scenario.

The way that the process works is to ramp the transfer by the specified step size in the base case. The base case state at the last solved ramped amount becomes the last solved state. Then all contingency scenarios are tested. If any contingency does not solve at the present transfer level, the contingency is studied individually to determine the transfer level at which it becomes critical. This transfer level will most likely be less than the last solved state transfer level.

Set current state as initial

Choose this option to set the initial state to be the current state. This might be useful if doing analysis outside of the PV tool and needing to be able to return to the present state. The **Restore initial state** option can then be used to return to the state designated as the initial state. This option might also be useful if desiring to do another analysis starting from the present state and choosing to do this by using the **Start over** option. The Start over option will restore back to the initial state before starting any analysis so it is important that the initial state be set correctly.

Keep in mind that once the **Run** button is clicked to start the analysis, whatever state is presently in place will be designated as the initial state.

Start over

This option will destroy any existing results, clear the log, clear all plots and plot controls, and restore the system state to the initial state. See comments in the **Restore initial state** and **Set current state as initial** options for more details on the initial state.

Run QV tool

This option will launch the QV analysis by opening the [QV Curves dialog](#) and then starting the analysis. The options specified on the [QV setup](#) tab will be used for the analysis.

Details about the sub-tabs at the bottom of page can be found at the following:

[Overview](#)

[Plot](#)

[Track Limits](#)

PV Curves Results: Overview

The Overview sub-tab is found on the [PV Results](#) tab of the [PV Curves dialog](#). This page provides a summary of all scenarios studied and their results. This table will update as the analysis progresses.

Overview						
Plot						
Track Limits						
	PV Scenario	Critical?	Critical Reason	Max Shift	Max Export	Max Import
1	base case	NO				
2	L_000001One-000002TwoC1	YES	Reached Nose	1162.50	1162.50	-898.26
3	L_000001One-000003ThreeC1	NO				
4	L_000002Two-000003ThreeC1	NO				
5	L_000002Two-000004FourC1	NO				
6	L_000002Two-000005FiveC1	NO				
7	L_000002Two-000006SixC1	YES	Reached Nose	675.00	675.00	-582.50
8	L_000003Three-000004FourC1	NO				
9	L_000004Four-000005FiveC1	NO				
10	L_000007Seven-000005FiveC1	YES	Reached Nose	2550.00	2550.00	-1452.39
11	L_000006Six-000007SevenC1	YES	Reached Nose	2537.50	2537.50	-1441.18
12	L_000006Six-000007SevenC2	YES	Reached Nose	2537.50	2537.50	-1441.18

The overview is a [case information display](#) and therefore can be used in a manner consistent with all other case information displays.

There are a number of fields that are shown by default on the display:

PV Scenario

Name of the scenario. Either *base case* or the contingency name.

Critical?

YES if the scenario is considered to be critical. Otherwise, *NO*.

Critical Reason

This will be blank as long as the scenario is not critical. If the scenario is critical, there are a number of reasons that could be displayed explaining why the scenario is critical:

Reached Nose - The power flow failed to solve so the system is considered to be at the nose of the PV curve.

Inadequate Voltage - The process stopped because at least one voltage is considered inadequate. This reason will only occur when using the option specified on the [Limit Violations](#) tab to **Stop when voltages become inadequate**. The message will be appended with the bus and voltage of the bus with the lowest inadequate voltage.

Source Maxed - The source no longer has enough resources to continue the power transfer. Source resources are affected by the options **Enforce unit MW limits** and **Do not allow negative loads** found on the [Common Options](#) sub-tab of the Setup tab. They could also be affected if choosing to use merit order dispatch in which unit limits are always enforced.

Sink Maxed - The sink no longer has enough resources to continue the power transfer. Sink resources are affected by the options **Enforce unit MW limits** and **Do not allow negative loads** found on the [Common Options](#) sub-tab of the Setup tab. They could also be affected if choosing to use merit order dispatch in which unit limits are always enforced.

SRT - Indicates that a solvable reverse transfer has been found when choosing to use the reverse transfer option found on the [Advanced Options](#) sub-tab of the Setup tab. This message is appended with the original critical reason that required the use of the reverse transfer.

RT Source Maxed - Indicates that the source no longer has enough resources to continue the power transfer when using the reverse transfer option found on the [Advanced Options](#) sub-tab of the Setup tab. This message is appended with the original critical reason that required the use of the reverse transfer.

RT Sink Maxed - Indicates that the sink no longer has enough resources to continue the power transfer when using the reverse transfer option found on the Advanced Options sub-tab of the Setup tab. This message is appended with the original critical reason that required the use of the reverse transfer.

Reached Nose in Base Case - This message will only occur for a contingency scenario. It indicates that the contingency scenario is actually less limiting than the base case scenario and that the analysis of the contingency scenario cannot continue because of the base case limitation. The power transfer is always ramped in the base case prior to implementing the contingency. The contingency cannot be attempted unless the base case solves with the transfer implemented.

Max Shift

This value will only be filled in for critical scenarios. This is the nominal shift that was implemented to reach the critical scenario. The nominal shift is the actual accumulated transfer that was attempted.

Note: With a few exceptions, shift values (includes Max Shift, Max Export, and Max Import) that are reported are from the last solved solution before the critical point was reached. Because the critical point is not considered a valid solution, no valid shift can be determined at this point. Therefore, the values that are reported are as close to this point as the process can get within the tolerances specified for the transfer step size. Exceptions to this occur when using the reverse transfer option. In this case, the values that are reported are for the minimum transfer level at which the contingency will solve and all voltages are adequate. Another exception occurs when a scenario becomes critical because of inadequate voltages at the zero transfer level. Because the scenario can be solved at this point, the shift values will be recorded at the actual step at which the voltages are inadequate. If a case becomes critical due to inadequate voltages and is not at the zero transfer level, the shift values reported for these will follow the rule that they come from the last solved step at which all voltages were adequate. The recording of values determined by [Quantities to Track](#) will also follow these same rules.

Max Export

This value will only be filled in for critical scenarios. This is the amount that the injection changed in the source injection group due to the transfer and contingency implementation. This value's magnitude can differ from the Max Shift value due to injection changes resulting from make-up power requirements from the contingency.

See note with Max Shift for additional explanation on how shift values are recorded.

Max Import

This value will only be filled in for critical scenarios. This is the amount that the injection changes in the sink injection group due to the transfer, loss changes due to the transfer, and contingency implementation. This value's magnitude can differ from the Max Shift value due to the injection changes resulting from make-up power requirements from the contingency and making up for losses due to the transfer. The sink injection group is designated to account for all losses due to the transfer.

See note with Max Shift for additional explanation on how shift values are recorded.

#Viol, Worst V Viol, Worst V Bus

These fields will be filled when choosing to monitor low or high voltage violations and there are violations for a given scenario. The options to monitor voltage violations are found on the [Limit violations](#) tab. These fields give the total number of all bus violations for a scenario, the voltage of the worst violation, and the bus at which the worst violation occurs, respectively. The worst violation is considered to be the one with the lowest voltage.

Max P Mism Bus #, Max P Mism Bus Name, Max MW Mism

These fields will only be filled in for critical scenarios. These give the bus number, bus name, and real power mismatch for the bus with the maximum real power mismatch for the power flow solution at which the scenario becomes critical. For scenarios that are at the nose of the PV curve, the mismatch should reflect the fact that the power flow solution will not converge and might give some insight into problem areas of the system.

Max Q Mism Bus #, Max Q Mism Bus Name, Max Mvar Mism

These fields will only be filled in for critical scenarios. These give the bus number, bus name, and reactive power mismatch for the bus with the maximum reactive power mismatch for the power flow solution at which the scenario becomes critical. For scenarios that are at the nose of the PV curve, the mismatch should reflect the fact that the power flow solution will not converge and might give some insight into problem areas of the system.

The right-click local menu has several options specific to PV results:

Show Dialog

Choosing this option will open the [Contingency Definition dialog](#) for the associated scenario's contingency. This dialog can be used to examine the contingency and make any necessary changes.

PV Curve records

There are two PV tool specific options available under this menu item:

Show Violations...

This option is available when choosing to monitor low or high voltage violations and there are violations for a given scenario. The options to monitor voltage violations are found on the [Limit violations](#) tab. When monitoring voltage violations, any bus voltages that violate limits will be stored for each scenario at the last studied transfer level. Choose Show Violations... for a given scenario to display a table of buses that have voltage violations.

Restore critical state

This option will restore the system state to the selected critical state. This means that the critical transfer and any associated contingency will be implemented. This option is only available if choosing to archive system states as set on the [PV output](#) tab. At a minimum the option to save critical states must be selected. Another condition is that the states must be saved as PowerWorld auxiliary (*.aux) files.

PV/QV Curves Results: Plot

Plotting functionality is almost identical for both the PV analysis and QV analysis (exceptions will be noted below). For PV analysis, the Plot tab is accessed from the [PV Results](#) tab of the [PV Curves dialog](#). For QV analysis, the Plots tab is accessed from the [QV Results](#) tab of the [QV Curves dialog](#). Any of the values selected to be [monitored](#) can be plotted using this display.

Horizontal axis value Pre-contingency
 Nominal Shift
 Vertical axis value type
 PU Volt
 Plot values for these elements ...

Description	Plot?
Three (3)	NO
Five (5)	NO

For these scenarios ... Always include base case

PV Scenario	Critical?	Plot?
base case	NO	NO
L_000001One-000002TwoC1	YES	NO
L_000001One-000003ThreeC1	NO	NO
L_000002Two-000003ThreeC1	NO	NO
L_000002Two-000004FourC1	NO	NO
L_000002Two-000005FiveC1	NO	NO
L_000002Two-000006SixC1	YES	NO
L_000003Three-000004FourC1	NO	NO
L_000004Four-000005FiveC1	NO	NO
L_000007Seven-000005FiveC1	YES	NO
L_000006Six-000007SevenC1	YES	NO

Plot title

The page is broken into two basic sections: the left and right half. The right half of the display allows selection of the scenarios that should be included on a given plot. The left half allows selection of the quantities that will be charted on the plot.

Selecting Scenarios to Plot

The right half of the display gives a table of all scenarios that were studied. Change the **Plot?** field to *YES* to include a scenario in the current plot. The Plot? field is a toggleable field and can be changed from *YES* to *NO* and back again by double-clicking on the appropriate entry. Any scenario whose Plot? field is set to *YES* will be included on the same plot once the **Plot** button is clicked.

The **Always include base case** box should be checked if the base case scenario should be included in the plot along with any other scenarios selected. This option is not available for QV results plotting.

Before a plot can be created, the quantities to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Selecting Quantities to Plot

The left half of the display gives the various options for selecting what quantities should be plotted.

The **Horizontal axis value** drop-down contains any parameters that were monitored. Additionally, for the PV analysis, this drop-down also contains the independent quantities (Nominal Shift, Export, and Import) recorded during the analysis. If the **Pre-contingency** box is checked, the x values for each plotted scenario will come from the base case instead of the results of a given scenario. The Pre-contingency check box is not available for QV results plotting.

The **Vertical axis value type** drop-down contains a list of all the types of values that were monitored. The type selected in this drop-down will determine the actual elements that are listed in the **Plot values for these elements** table. To include an element in the plot, change the **Plot?** field to *YES* for that element. The Plot? field is a toggleable field and can be changed from *YES* to

NO and back again by double-clicking on the appropriate entry. Any element whose Plot? field is set to *YES* will be included in the same plot once the **Plot** button is clicked.

Before a plot can be created, the scenarios to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Plot Title

Optional field that can be set to add a title to a plot and distinguish one plot from another.

Plot

After selecting the scenarios and values to plot, click this button to actually generate the plot. The plot will appear in its own window. The number of plots that can be created is limited only by computer system memory.

Right-clicking on a plot will expose a local menu with several options. The plot can be saved as a bitmap, Windows metafile, JPEG, or text file by clicking **Save** and selecting the appropriate file type. **Copy** will allow the plot to be copied and pasted into other applications. **Print** will send the plot to a printer that can be configured by selecting the **Printer Setup** option. Selecting **Close** will close the plot window.

Clear

Click this button to reset all of the **Plot?** fields to *NO* for both the scenarios to plot and the values to plot.

PV/QV Curves Results: Track Limits

The Track Limits sub-tab is found on the [PV Results](#) tab of the [PV Curves dialog](#) or the [QV Results](#) tab of the [QV Curves dialog](#). The results displayed are similar for both the PV and QV analysis. This page provides access to the results of [tracking devices at limits](#) which are determined in the same manner for both the PV and QV analysis.

Overview | Plot | Track Limits

Filter out devices that never hit or backoff a limit during the PV run

Generator Var Limits | Shunt Var Limits | LTC Var Limits | Line Thermal Limits | Interface Thermal Limits

Records ▾ Set ▾ Columns ▾ SORT BY RECD Options ▾

	Number of Bus	Name of Bus	ID	0.00	100.00	200.00	225.00	250.00	300.00
1	1	One	1	Within Range	At Min	At Min	Within Range	Within Range	Within Range
2	2	Two	1	Within Range	Within Range	At Max	At Max	At Max	At Max
3	4	Four	1	Within Range					
4	6	Six	1	Within Range	At Max				
5	7	Seven	1	Within Range	Within Range	At Max	At Max	At Max	At Max

The page is divided into several additional sub-tabs for the different types of devices that can be tracked. Tables on each tab are similar in that they list all of the devices of that type that were tracked. For PV analysis, each column of the table is the transfer amount at which the device limits were checked. For QV analysis, each column of the table is the per-unit voltage set-point for the study bus during the scenario in which the limits were checked. The results show where each element was located relative to its min/max limits during each step of the analysis using the indicators of *Within Limits*, *Within Range*, *At Max*, and *At Min*. To simplify the display and exclude those elements that were always within their limits during all steps of the process, check the box **Filter out devices that never hit or backoff a limit during the PV run**.

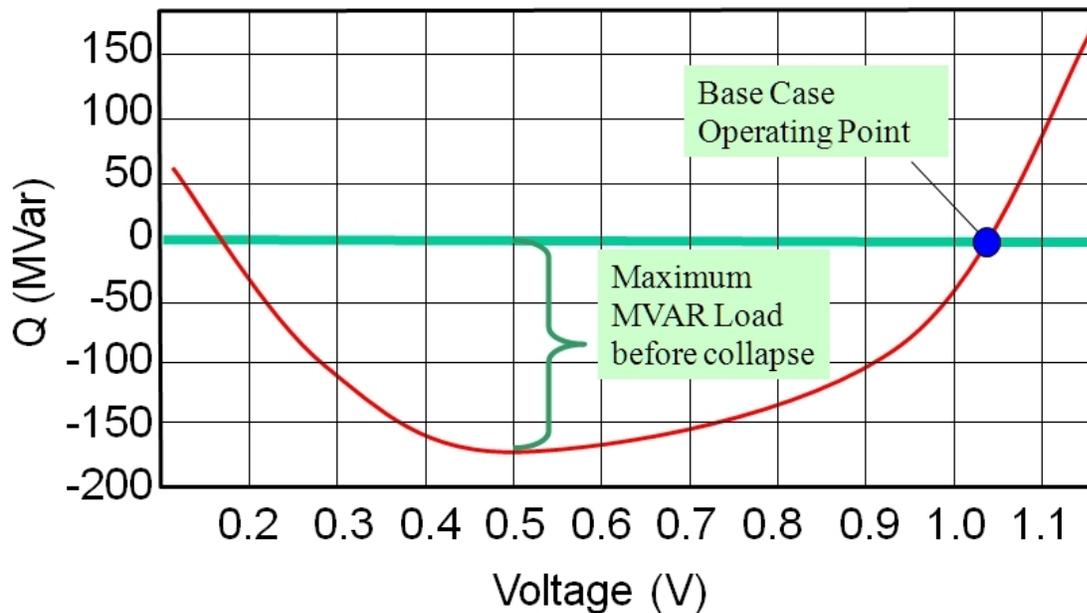
QV Curves

The QV Curves tool provides the ability to compute QV curves for any bus in the system. A QV analysis studies how variations in reactive power (Q) injection at a bus affects the voltage (V) at that same bus. Other system parameters can also be monitored as the reactive power injection changes.

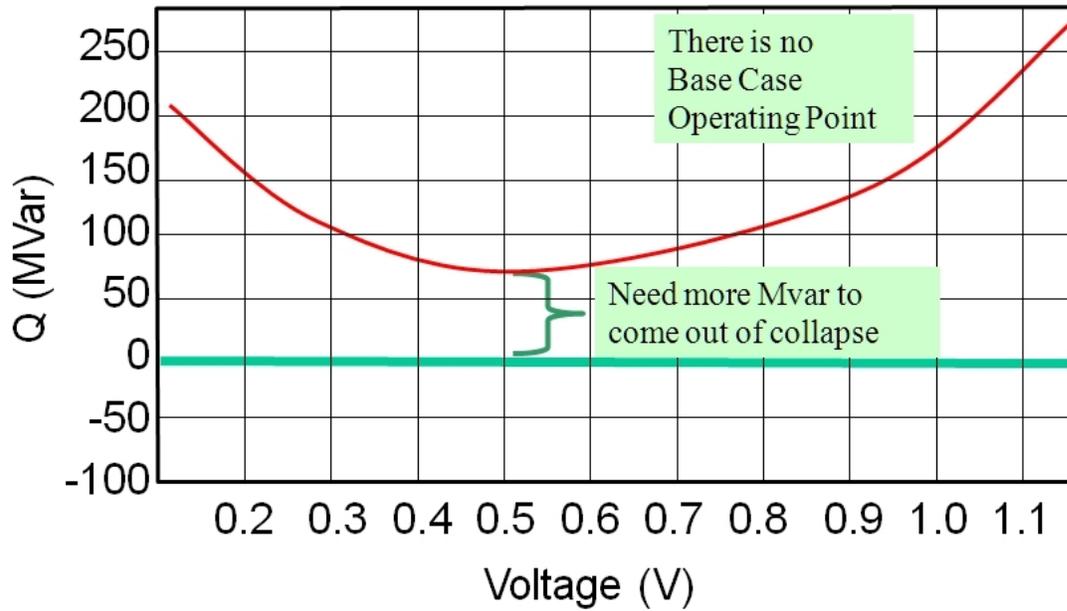
To create a QV curve, a fictitious generator (synchronous condenser) is placed at a bus that is being studied. The voltage set-point of this generator is varied and its var output is allowed to be any value needed to meet this voltage set-point. The vertical axis (y-axis) of a QV curve depicts the output of the fictitious generator in Mvar. The horizontal (x-axis) depicts the respective voltage under this condition. The base case operating point of the system is represented by the x-intercept of the curve. This is the point where the fictitious generator is at 0 Mvar output and represents the base case. There are situations in which the output of the fictitious generator is not 0 Mvar in the base case, but these will be explained with the appropriate option settings. **When considering a contingency scenario in the QV analysis, the term *base case operating point* will also be used. This is the starting case that represents the system prior to performing any analysis for the curve tracing.**

Tracing down the curve from higher to lower voltage set-points represents a decrease in the fictitious generator's Mvar output which is representative of an increase in Mvar load. The curve is then tracing what the voltage would be as the Mvar load increases. At some point the Mvar value of the generator will stop decreasing and the bottom of the curve will be reached. This point represents the maximum increase in load Mvar that can occur at this bus before voltage collapse is reached.

The following shows a typical QV curve. The plot is actually a VQ curve, but this is traditionally called the QV curve.



The following plot shows a situation in which there is not enough Mvar reserve. There is no base case operating point as the curve never crosses the x-axis. The difference between the bottom of the curve and the x-axis is the amount of Mvar injection needed to achieve a solvable case and to come out of collapse.



To use the QV Curves tool, select **QV Curves** on the **PV and QV Curves (PVQV)** ribbon group on the [Add Ons](#) ribbon tab. The [QV Curves dialog](#) will open from which buses to be monitored can be specified, options for the QV curves can be set, and the QV curve analysis can be run.

QV Curves: Buses

The Buses tab is found on the [QV Curves dialog](#).

Buses							
Quantities to track		Options		Results			
	Number	Name	Nom kV	Selected?	Min Volt	Max Volt	Step Size
1	1	One	138.00	NO			
2	2	Two	138.00	NO			
3	3	Three	138.00	NO			
4	4	Four	138.00	NO			
5	5	Five	138.00	NO			
6	6	Six	138.00	NO			
7	7	Seven	138.00	NO			

To add buses to the QV analysis, specify ...

A range of bus numbers

or the name of a bus

Additionally, automatically draw curves for

lowest-voltage buses and

highest dv/dq buses

from limit group

The Buses page is used to designate the buses at which QV curves should be calculated. Depending on other options that are selected, a QV curve will be determined for each selected bus for the base case and any selected contingencies. Each bus/contingency (or base case) pair comprise a QV scenario.

By default, the table lists all buses in the case. This table is a bus case information display and [filtering](#) can be applied to this table to make the selection of buses easier. Several QV-specific fields are listed by default:

Selected?

Set this field to *YES* to calculate a QV curve for this bus. This field can be toggled between *YES* and *NO* by double-clicking on the appropriate entry.

Min Volt

Specify the minimum voltage at which the analysis will be performed for this particular bus. If this is not specified, the default minimum voltage set with the default [options](#) is used. Only non-default values will be shown. If the value is changed to the default setting, the cell will be set blank.

Max Volt

Specify the maximum voltage at which the analysis will be performed for this particular bus. If this is not specified, the default maximum voltage set with the default [options](#) is used. Only non-default values will be shown. If the value is changed to the default setting, the cell will be set blank.

Step Size

Increment between voltage set-points used to trace the curve for this particular bus. If this is not specified, the default step size set with the default [options](#) is used. Only non-default values will be shown. If the value is changed to the default setting, the cell will be set blank.

Buses can also be selected for analysis by using several options found at the bottom of the dialog. **A range of bus numbers or the name of a bus** can be specified in the appropriate boxes. Clicking the **Add** button for either entry will set the **Selected?** field to *YES* for the designated buses. These options are used to make setting the Selected? field more convenient.

There is another set of options that will have the QV tool determine which buses should be analyzed in addition to those already selected.

Additionally, automatically draw curves for

Check this box to optionally draw additional curves based on certain conditions. Set the number of **lowest-voltage buses** and/or **highest dv/dq buses** at which curves should be traced. The dv/dq sensitivity is the sensitivity of voltage at a bus to an injection of reactive power at the same bus. The sensitivities are ranked based on their magnitude. These lowest-voltage buses and highest dv/dq buses are determined for each scenario once any contingency has been applied. Use the **from limit group** drop-down to specify which buses should be considered when evaluating these conditions. Only buses that are assigned to the selected [limit group](#) will be considered.

If the buses identified when evaluating these conditions already have been selected by the user, no additional buses will be selected to make sure that the number of additional buses to include is achieved by buses that have not been user selected.

QV Curves: Options

The Options tab is found on the [QV Curves dialog](#). The Options page contains several additional sub-tabs from which parameters necessary for carrying out a QV analysis can be set:

[Solution](#)

[Contingencies](#)

[Output](#)

The **Restore Defaults** button is available on the Options tab regardless of the sub-tab that is selected. Clicking this button will set all options found on any of the sub-tabs to the default settings.

QV Curves Options: Solution

These options are found on the Solution sub-tab located on the [Options](#) tab of the [QV Curves dialog](#).

Default solution parameters

Voltage step size (pu)

Increment between voltage set-points used to trace a QV curve. This is the default value that will be applied to a bus unless a bus-specific value is specified.

Minimum voltage (pu)

Minimum voltage at which the analysis will be performed. This is the default value that will be applied to a bus unless a bus-specific value is specified.

There are three ways to set the minimum voltage that will be used for a particular bus. The default value will be used unless a bus-specific value is specified. A third option is possible if launching the QV analysis upon finishing the PV analysis and choosing to use QV accelerator settings. If using the accelerator settings, the bus-specific minimum voltage will be set based on the accelerator settings. More information is available about launching the QV analysis from the PV analysis and setting the accelerator settings on the [QV Setup](#) tab of the [PV Curves dialog](#).

Maximum voltage (pu)

Maximum voltage at which the analysis will be performed. This is the default value that will be applied to a bus unless a bus-specific value is specified.

There are three ways to set the maximum voltage that will be used for a particular bus. The default value will be used unless a bus-specific value is specified. A third option is possible if launching the QV analysis upon finishing the PV analysis and choosing to use QV accelerator settings. If using the accelerator settings, the bus-specific maximum voltage will be set based on the accelerator settings. More information is available about launching the QV analysis from the PV analysis and setting the accelerator settings on the [QV Setup](#) tab of the [PV Curves dialog](#).

Use initial voltage as Vmax

Check this box to use the initial voltage at a bus as the maximum voltage at which the analysis will be performed. This option overrides any of the other options that set the maximum voltage for a bus. The initial voltage at a bus is determined prior to starting the QV analysis but after applying contingencies and applying any additional var support necessary to make an unsolvable scenario solvable.

Power flow solution options

Global solution options

Click this button to bring up the [Simulator Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases and the options used when no contingency-specific solution options are defined. Changing these options affect all power flow computations, even those outside the QV process.

The global solution options will be used in the QV analysis unless QV-specific solution options are defined using the option described below. If contingency-specific solution options are defined for a contingency, those options will override both the QV-specific solution options and the global solution options.

QV Power Flow Solution Options

Click this button to bring up a dialog that allows specification of QV-specific power flow solution options. These are the same options that are described in the Simulator Options Dialog for [Power Flow Solution Options](#). The QV-specific solution options will override the global solution options. If contingency-specific solution options are defined for a contingency, those options will override both the QV-specific solution options and the global solution options.

Because the goal is to stress the system while performing the QV analysis, there are two power flow solution options that are set internally as part of the QV process. These are the options that specify the minimum per unit voltage for constant power and constant current loads. These options are both set to 0 pu and override any user-specified settings.

Minimum Per Unit Voltage for	
Constant Power Loads	0.000
Constant Current Loads	0.000

These options are both set to 0 pu during both the base case solution and any contingency case solution.

QV Curves Options: Contingencies

These options are found on the Contingencies sub-tab located on the [Options](#) tab of the [QV Curves dialog](#).

The QV curve utility allows you to compute QV curves for each of the buses you have selected for the base case and for any number of contingencies. The contingencies you want to process must have already been defined using Simulator's Contingency Analysis feature. Use the checkbox below to specify whether you want to compute QV curves for the presently defined contingencies. If you leave this box unchecked, QV curves will be computed only for base case conditions.

Process each of the currently defined contingencies

Skip base case

Attempt to make unsolvable contingencies solvable with synchronous condenser

Restore Defaults

Process each of the currently defined contingencies

Check this box to have contingencies included in the QV analysis.

QV curves can be calculated for the specified buses for both base and contingency conditions. To analyze a set of contingencies, these must be defined using the separate [Contingency Analysis Dialog](#). Any contingency whose **Skip** field is set to *NO* will then be included in the QV analysis.

Another option that is relevant to the QV analysis that must be set for a contingency on the contingency analysis dialog is the **QV Autoplot** field. This field must be set to *YES* in order to have a QV curve plotted automatically for a contingency scenario. More information about the option to plot curves automatically is found on the [Output](#) sub-tab.

Contingencies for inclusion with the QV analysis can also be specified based on the results of the PV analysis if choosing to launch the QV analysis immediately following the PV analysis. More information on launching the QV analysis upon completion of the PV analysis is found on the [QV Setup](#) tab on the [PV Curves dialog](#). Even when specifying contingencies based on the PV analysis, this check box must be checked in order to actually include the contingencies with the QV analysis.

Note: The Robust Solution Process option that can be utilized with the contingency analysis when a contingency solution failure occurs is not applied when a contingency is implemented during the QV analysis. All other options defined with contingency analysis such as contingency-specific solution options and make-up power specifications are used when applying a contingency during the QV analysis.

Skip base case

Check this box to prevent the computation of QV curves for base case conditions.

Attempt to make unsolvable contingencies solvable with synchronous condenser

Some contingencies may not solve when implemented. If this box is checked, an attempt will be made to make an unsolvable contingency solvable by providing additional var injection/absorption through use of a fictitious generator (synchronous condenser). As each selected bus is processed, the fake generator (synchronous condenser) that is normally added to trace the QV curve at a bus, will first be used to determine if Mvar injection/absorption can be adjusted such that a solvable point is found. If so, this solvable point with the fake generator injection/absorption in place will then be the starting point for the QV curve tracing.

The **Q0** results field found on the [Listing](#) sub-tab of the [Results](#) tab indicates the injection of the fake generator at the base case or initial contingency solution point. This field being non-zero is indicative of an unsolvable contingency with the value of the field being the Mvar injection required to solve the contingency.

Finding a solvable point can be time consuming if there are a number of contingencies that do not solve and a large number of buses are selected for analysis. Uncheck this box if long solution times become an issue.

QV Curves Options: Output

These options are found on the Output sub-tab located on the [Options](#) tab of the [QV Curves dialog](#).

Save results in comma-delimited format

A comma-delimited file format for QV results exists for easy import into a spreadsheet. This file format cannot be loaded back into Simulator. If this functionality is required, it is suggested that the results be saved in an auxiliary file. To save the results to an auxiliary file, click the **Save...** button found at the bottom of the QV Curves dialog. The values saved to the comma-delimited file are the same values that are saved in the SUBDATA section for QVCurve data saved to an auxiliary file.

The comma-delimited file is grouped into sections for each scenario. Each scenario is identified by a header that indicates the bus and either base case or contingency identifier. The header is followed by a line that provides the field order of the actual data. The field line is followed by lines of data. Each scenario grouping is separated by a blank line.

The fields have the following meaning:

V(PU) - per-unit voltage set-point

Q_sync(MVR) - output of the fake generator (synchronous condenser)

Q_shunt(MVR) - any existing on-line switched shunt output

Q_tot(MVR) - $Q_sync + Q_shunt$

Q_res(MVR) - available reserve at the bus

Q_tot_res(MVR) - $Q_sync + Q_shunt + Q_res$

Here is an example of the comma-delimited file:

```

**BUS** 3 (Three_138.0),**CASE** L_000001ONE-000003THREEC1
V(PU), Q_sync(MVR), Q_shunt(MVR), Q_tot(MVR), Q_res(MVR), Q_tot_res(MVR)
1.1000, 437.5849, 48.4000, 485.9849, 242.0000, 727.9849
1.0962, 419.8296, 48.0627, 467.8923, 240.3136, 708.2059
1.0862, 374.1186, 47.1898, 421.3083, 235.9489, 657.2572
1.0762, 329.2408, 46.3248, 375.5656, 231.6243, 607.1899
1.0662, 285.1944, 45.4679, 330.6623, 227.3396, 558.0020
1.0562, 241.9775, 44.6190, 286.5965, 223.0950, 509.6915
1.0462, 199.5884, 43.7781, 243.3665, 218.8904, 462.2568
1.0362, 158.0253, 42.9451, 200.9705, 214.7257, 415.6962
1.0262, 117.2866, 42.1202, 159.4069, 210.6011, 370.0079
1.0162, 77.3706, 41.3033, 118.6739, 206.5164, 325.1904
1.0062, 38.2757, 40.4944, 78.7701, 202.4718, 281.2419
0.9962, 0.0000, 39.6934, 39.6934, 198.4671, 238.1605
0.9962, 0.0000, 39.6934, 39.6934, 0.0000, 39.6934
0.9962, 0.0000, 39.6934, 39.6934, -119.0803, -79.3869
0.9862, -37.4572, 38.9005, 1.4433, -116.7015, -115.2582
0.9762, -74.0981, 38.1156, -35.9825, -114.3467, -150.3292
0.9662, -109.9612, 37.3386, -72.6225, -112.0160, -184.6385
0.9562, -144.9682, 36.5697, -108.3985, -109.7092, -218.1077
0.9462, -179.1652, 35.8088, -143.3564, -107.4264, -250.7828
0.9362, -212.5517, 35.0559, -177.4958, -105.1676, -282.6635

```

Save Results to File

Check this box to actually save the results to the file specified by **Save results in file**.

Save results in file

Specify the name and file path of the file to which to save the results. This option will be enabled when the **Save Results to File** option is checked. Click the **Browse** button to open a dialog that will allow selection of the file path and name.

Plot curves as they are computed

Check this box to draw and update QV curve plots during each step of the QV analysis process. In order to have QV curves automatically plotted for a contingency during each step of the analysis, the **QV Autoplot** field belonging to that contingency must be set to *YES* on the [Contingency Analysis Dialog](#).

If curves are not plotted at they are computed, they can be easily created upon completion of the analysis from an option found on the [Listing](#) sub-tab on the [Results](#) tab.

When plotting V versus Q, treat Q as...

There are several options that determine how the Q quantity should be plotted on a QV curve. These options are applicable whether plotting the curves automatically as they are computed or plotting the curves upon completion of the analysis.

An example plot is shown below with examples of the plots that result from the four possible options for how to treat Q.

The output of the fictitious synchronous condenser (Qsync)

Select this option to treat Q as the output of the fake generator (synchronous condenser) only. This is the Q_sync(MVR) curve shown in the example plot below.

The total reactive injection at the bus, including shunts but excluding load

Select this option to treat Q as the sum of the output of the fake generator (synchronous condenser) and any existing on-line shunts at the bus. Shunts include both bus shunts and switched shunts. This is the Q_tot(MVR) curve shown in the example plot below.

Include reserves (status = closed, generators on AVR, switched shunts not fixed)

Check this box to offset the Q values by the appropriate Mvar reserves values.

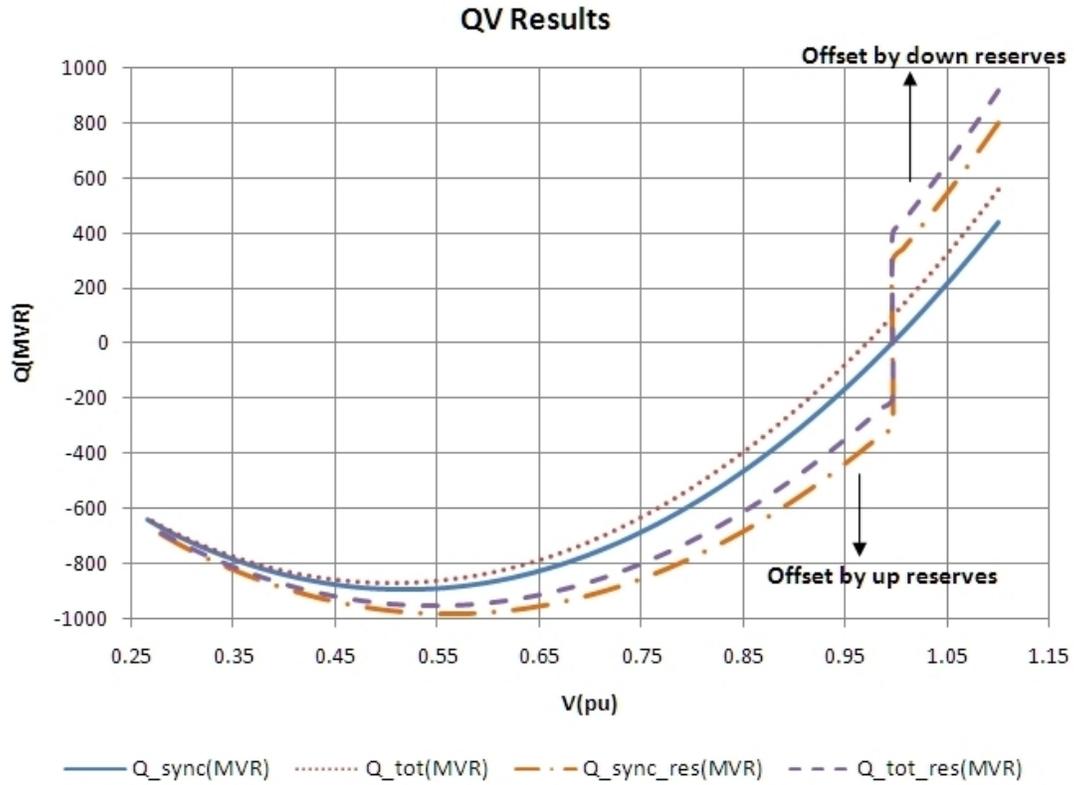
When the QV curve is traced for a given bus, any existing switched shunts on control at that bus are turned off control and any existing generators on AVR control at that bus are turned off control. The amount of Mvar support that these devices can provide is then recorded and determined at each voltage set-point. These are the reserves values. Only devices that are on-line and on control will be included. This means that for a generator the AVR status = *YES* and for a switched shunt the control mode must not be *Fixed*.

The amount of down (decrease) reserves at a given voltage set-point for a switched shunt is determined by the difference between the current nominal output of the shunt and its minimum nominal output multiplied by the square of the voltage set-point: $(\text{Nominal Mvar} - \text{Min Nominal Mvar}) * V^2$. The amount of down (decrease) reserves for a generator at a given voltage set-point is the difference between the current output and the minimum Mvar output: $(\text{Mvar Output} - \text{Min Mvars})$. The amount of up (increase) reserves at a given voltage set-point for a switched shunt is determined by the difference between the maximum nominal output of the shunt and its current nominal output multiplied by the square of the voltage set-point: $(\text{Max Nominal Mvar} - \text{Nominal Mvar}) * V^2$. The amount of up (increase) reserves for a generator at a given voltage set-point is the difference between the maximum Mvar output and the current output: $(\text{Max Mvars} - \text{Mvar Output})$. The total reserves of each type at any voltage set-point is the sum of reserves of that type for all qualifying switched shunts and generators at the studied bus.

Which type (up or down) of reserves to consider at a particular voltage set-point is determined based on the location of the voltage set-point relative to the base case operating voltage. When considering contingencies, the base case operating voltage is the voltage at a bus following implementation of the contingency but before any QV curve tracing is started. If the voltage set-point is less than the base case operating voltage, the up reserves are considered. This part of the curve represents an increase in the Mvar load at a bus. The difference in the bottom of the curve and the x-axis is the amount that the Mvar load at a bus can be increased before voltage collapse. Being able to inject additional Mvar through available reserves will further increase this margin. When plotting including reserves, the portion of the curve to the left of the base case operating point is then offset downward by the amount of up reserves that are available at each voltage set-point, thus increasing the margin before collapse. The portion of the curve to the right of the base case operating point is then offset upward based on the amount of down reserves available at each voltage set-point.

Instead of listing an absolute value of reserves available, the Q_res(MVR) value as reported with the QV output results (either comma-delimited file or auxiliary file) is reflective of the offset required when plotting the curve including the reserves.

When choosing to include reserves, the values that are plotted are either Q_sync_res(MVR) or Q_tot_res(MVR). The Q_sync_res(MVR) value is not included in the QV output results (either comma-delimited file or auxiliary file) but is equal to the sum of Q_sync(MVR) and Q_res(MVR): $Q_sync_res(MVR) = Q_sync(MVR) + Q_res(MVR)$. Q_sync_res(MVR) represents the combination of selecting the option to plot **The output of the synchronous condenser** and including reserves. Q_tot_res(MVR) represents the combination of selecting the option to plot **The total reactive injection at the bus...** and including reserves.



Note: The reserves and the existing injection in the above plot are represented solely by switched shunts. All of the curves tend to converge toward the same Q at lower voltages. This is because of the relationship of the actual output of a switched shunt to the nominal output: (Actual Mvar) = (Nominal Mvar)*V².

PV/QV Quantities to Track

Specification of quantities to track during analysis is almost identical for both the PV and QV analysis (exceptions will be noted below). For PV analysis, the Quantities to Track tab is accessed from the [PV Curves](#) dialog, and for the QV analysis, this tab is accessed from the [QV Curves](#) dialog. Quantities to track are monitored and stored as the transfer increases for the PV analysis or the voltage set-point changes for the QV analysis. Any unselected system parameters will not be saved. The Quantities to Track page contains several sub-tabs that allow the monitoring of different types of objects including: buses, generators, injection groups, lines, transformers, shunts and interfaces. There is also a sub-tab for monitoring devices at limits.

Buses

Voltage - per unit voltage

kV Voltage - actual voltage in kV

Angle - voltage angle in degrees

MW Load - total real power load

Mvar Load - total reactive power load

dV/dQ - sensitivity of voltage to change in reactive power injection at the same bus

VP Sensitivity - sensitivity of voltage change due to real power transfer. For PV analysis this is the sensitivity of the voltage at the bus due to the selected transfer from source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Shunt Inj - total switched shunt reactive power (Mvar) output

The default value of all bus quantities is *NO*, indicating that the analysis will not monitor any bus-related quantities. To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

For PV analysis if the intent is to trace PV curves, it makes sense to monitor at least one bus voltage. The process will not automatically monitor and store voltage quantities.

For QV analysis, per unit voltage will be monitored automatically for all buses selected to be included in the QV analysis.

Generators

Gen MW - real power output

Gen Mvar - reactive power output

Mvar Reserve - reactive power reserves in the positive direction (Max Mvar - Mvar)

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

Groups

This allows the monitoring of injection group parameters.

Mvar Reserve - total reactive power reserves in the positive direction for all generators in the group (Max Mvar - Mvar)

Gen MW - total real power output of all generators in the group

Gen Mvar - total reactive power output of all generators in the group

Load MW - total real power load for all loads in the group

Load Mvar - total reactive power load for all loads in the group

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

Lines

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab.

Branch MW - real power flow on the branch in either the FROM-TO or TO-FROM direction

Branch Mvar - reactive power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MVA - total power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MW Loss - real power loss

Branch Mvar Loss - reactive power loss

Branch PTDF - [Power Transfer Distribution Factor](#) on the branch due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Xfmr Tap - tap ratio if the branch is a transformer

For the flow (Branch MW, Branch Mvar, and Branch MVA) and Branch PTDF fields, double-clicking on a particular entry will toggle its value from *NO* to *FROM-TO*, and double-clicking again will toggle its value to *TO-FROM*. Double-clicking on a loss field or Xfmr Tap field will toggle the value between *YES* and *NO*.

Transformers

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab. See **Lines** above for setting up common branch quantities. Only branches containing transformers will appear on the **Transformers** sub-tab.

Type - set the transformer type (Fixed, LTC, Mvar, or Phase)

Reg Val - regulated value for the transformer control

Tap Pos - tap position

Reg Err - deviation of the regulated value from the regulation Min/Max range

All fields specified above except for the Type field are used to determine which quantities should be monitored and stored during the analysis. The Type field can be accessed here in order to change the type of transformer. The type will not be monitored during the analysis.

Shunts

This allows the monitoring of switched shunt parameters.

Actual Q - actual reactive power output

Actual P - actual real power output

Nom Q - nominal reactive power output

Nom P - nominal real power output

Reg Err - deviation of the regulated value from the regulation Min/Max range. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

Reg Val - value of regulated quantity at the regulated bus. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

Interfaces

MW Flow - real power flow

Mvar Flow - reactive power flow

MVA Flow - total power flow

MW Loss - real power losses

Mvar Loss - reactive power losses

Interface PTDF - [Power Transfer Distribution Factor](#) on the interface due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

ATC Extra Monitors

This option is only available when using the PV Curves tool. The following will appear at the bottom of the Quantities to Track tab accessed from the PV Curves dialog.

Include ATC Extra Monitors

Define ATC Extra Monitors

[ATC Extra Monitors](#) are typically used with the [ATC add-on tool](#) to monitor special branches or interfaces in addition to ones monitored by setting the [Limit Monitoring Settings](#). This would be done for example if trying to determine an interface flow limit so that interface can be used as a proxy for limiting the transfer between a source and a sink. This same functionality is desirable when using the PV analysis. Both the ATC and PV analyses are similar in that they study the impacts of real power transfers. The ATC analysis is concerned with thermal limitations while the PV analysis looks at voltage stability limitations. Use of the ATC Extra Monitors with the PV analysis could be additionally useful for developing a proxy interface to help prevent voltage stability problems.

Click the **Define ATC Extra Monitors** button to view a list of existing ATC Extra Monitors or create new ones. To include monitoring of these elements at part of the PV analysis, check the **Include ATC Extra Monitors** box.

When monitoring ATC Extra Monitors with the PV analysis, the real power (MW) flow on the specified branches and/or interfaces will be monitored at each base case transfer level. No monitoring is done with contingencies implemented.

ATC Extra Monitors can be added to the Overview table, available on the [Results](#) tab, via the [Display/Column Options...](#) dialog. When displayed in this table, the value reported with the extra monitor is the flow on the extra monitor at the Max Shift level of transfer for the given scenario. Values for ATC Extra Monitors will only be displayed in the Overview table for critical scenarios.

Access to the ATC Extra Monitor values stored at transfer levels other than the Max Shift transfer for critical scenarios and values stored for non-critical scenarios is available through SUBDATA sections of the PWPVResultListContainer data type that is saved when saving PV results to an auxiliary file or in the output file saved when choosing to save results to file from the [PV output](#) tab.

Devices at Limits

The **Devices at Limits** sub-tab allows the selection of various options to track devices that hit or back off limits during the PV analysis. Generators and switched shunts at var limits, LTC transformers at tap limits, and lines and interfaces at thermal limits can all be tracked.

Track Hitting and Backing Off Limits during Ramping

<input type="checkbox"/> Generators Var Limits	<input type="text"/>	<input type="button" value="Define..."/>
<input type="checkbox"/> Switched Shunts Var Limits	<input type="text"/>	<input type="button" value="Define..."/>
<input type="checkbox"/> LTC Transformers Tap Limits	<input type="text"/>	<input type="button" value="Define..."/>
<input type="checkbox"/> Lines Thermal Limits	<input type="text"/>	<input type="button" value="Define..."/>
<input type="checkbox"/> Interfaces Thermal Limits	<input type="text"/>	<input type="button" value="Define..."/>

Note: Narrow limit tracking by specifying a filter
 Note: If no filter is specified, then all objects will be tracked.

To track the limits of any of these devices, check the appropriate checkbox. Next to each checkbox there is a dropdown box for selecting an [advanced filter](#) to apply to the tracking. A new filter can be defined for a particular device by clicking the **Define** button next to that device. The filter will limit the number of devices that get tracked. It is a good idea to define a filter for tracking devices so that all elements in the case will not be tracked. Keep in mind that tracking any quantities will take up space in the computer memory. If too many devices are tracked, it is possible to run out of memory.

With these options, device limits are only tracked in the base case at valid transfer levels. No device limit tracking is done with contingencies implemented.

QV Curves: Results

The Results tab is found on the [QV Curves dialog](#). This page provides access to all of the QV results. This page is divided into several additional sub-tabs from which different categories of results are available:

[Listing](#)

[Plots](#)

[Track Limits](#)

The Results tab also displays messages about the progress of the analysis. The messages are displayed regardless of the sub-tab that is selected.

QV Curves Results: Listing

The Listing sub-tab is found on the [QV Results](#) tab of the [QV Curves dialog](#). This page provides a summary of all scenarios studied and their results. This table will update as the analysis progresses.

V = 0.1049 Computing Q for bus Five, Contingency L_000007Seven-000005FiveC1											
Number	Name	Nom kV	Case Name	V at Q0	Q0	Qinj_0	Vmax	Q at VMax	Qinj at Vmax	V a	
1	5 Five	138.00	BASECASE	0.95	0.00	0.00	1.1000	191.51	191.51		
2	5 Five	138.00	L_000001ONE-000002TWOC1	0.96	0.00	0.00	1.1000	191.14	191.14		
3	5 Five	138.00	L_000001ONE-000003THREEC1	0.95	0.00	0.00	1.1000	192.14	192.14		
4	5 Five	138.00	L_000002TWO-000003THREEC1	0.95	0.00	0.00	1.1000	192.22	192.22		
5	5 Five	138.00	L_000002TWO-000004FOURC1	0.95	0.00	0.00	1.1000	192.30	192.30		
6	5 Five	138.00	L_000002TWO-000005FIVEC1	0.95	91.68	91.68	1.1000	159.12	159.12		
7	5 Five	138.00	L_000002TWO-000006SIXC1	0.00	0.00	0.00	0.0000	0.00	0.00		
8	5 Five	138.00	L_000003THREE-000004FOURC1	0.96	0.00	0.00	1.1000	191.12	191.12		
9	5 Five	138.00	L_000004FOUR-000005FIVEC1	0.92	0.00	0.00	1.1000	151.38	151.38		
10	5 Five	138.00	L_000007SEVEN-000005FIVEC1	0.95	0.00	0.00	1.1000	191.51	191.51		

The listing is a [case information display](#) and therefore can be used in a manner consistent with all other case information displays.

There are a number of fields that are shown by default on the display:

Number, Name, Nom kV

Number, name, and nominal kV voltage of the studied bus, respectively. This along with the Case Name identifies the scenario.

Case Name

Either *BASECASE* or the name of a contingency. This along with the **Number**, **Name**, and **Nom kV** identifies the scenario.

V at Q0

Per-unit voltage in the starting case (base case operating point). The starting case is either the base case or a contingency case. This is the voltage that is recorded prior to starting any QV curve tracing. This is the voltage at **Q0**.

If the case is not solvable and a successful attempt is made at finding the solvable point by using the fake generator to provide Mvar injection, this is the voltage recorded once the case becomes solvable.

Q0

Mvar output of the fake generator in the starting case (base case operating point). The starting case is either the base case or contingency case. This is the Mvar output that is recorded prior to starting any QV curve tracing.

If the case is not solvable and a successful attempt is made at finding the solvable point by using the fake generator to provide Mvar injection, this is the Mvar output of the fake generator that is required to make the case solvable. Therefore, a non-zero entry means that the initial starting case was not solvable.

Qinj_0

Total Mvar injection in the starting case (base case operating point). The starting case is either the base case or contingency case. This is the total injection that is recorded prior to starting any QV curve tracing. This includes the shunt injection from bus shunts and switched shunts and the output of the fake generator.

Vmax

Maximum per-unit voltage that was recorded.

Q at VMax

Mvar injection of the fake generator at the maximum per-unit voltage that was recorded, **Vmax**.

Qinj at Vmax

Total Mvar injection at the maximum per-unit voltage that was recorded, **Vmax**. This includes the shunt injection from bus shunts and switched shunts and the output of the fake generator.

V at Qmin

Per-unit voltage at the minimum recorded fake generator injection, **Qmin**.

Qmin

Minimum recorded fake generator injection. If negative, this is the amount that the Mvar load can be increased before voltage collapse. If positive, this is the amount of Mvar injection required to come out of collapse and achieve a solvable case.

Qinj_min

Total minimum recorded Mvar injection. This includes the shunt injection from bus shunts and switched shunts and the output of the fake generator.

Vmin

Minimum per-unit voltage that was recorded.

Q at Vmin

Mvar injection of the fake generator at the minimum per-unit voltage that was recorded, **Vmin**.

Qinj at Vmin

Total Mvar injection at the minimum per-unit voltage that was recorded, **Vmin**. This includes the shunt injection from bus shunts and switched shunt and the output of the fake generator.

The right-click local menu has one option specific to QV results:

Plot QV Curve

This will plot the QV curve for the selected scenario. What value of Q to plot is determined by the options selected for how to treat Q as found on the [Output](#) sub-tab of the [Options](#) tab.

PV/QV Curves Results: Plot

Plotting functionality is almost identical for both the PV analysis and QV analysis (exceptions will be noted below). For PV analysis, the Plot tab is accessed from the [PV Results](#) tab of the [PV Curves dialog](#). For QV analysis, the Plots tab is accessed from the [QV Results](#) tab of the [QV Curves dialog](#). Any of the values selected to be [monitored](#) can be plotted using this display.

Horizontal axis value Pre-contingency
 Nominal Shift
 Vertical axis value type
 PU Volt
 Plot values for these elements ...

Description	Plot?
Three (3)	NO
Five (5)	NO

For these scenarios ... Always include base case

PV Scenario	Critical?	Plot?
base case	NO	NO
L_000001One-000002TwoC1	YES	NO
L_000001One-000003ThreeC1	NO	NO
L_000002Two-000003ThreeC1	NO	NO
L_000002Two-000004FourC1	NO	NO
L_000002Two-000005FiveC1	NO	NO
L_000002Two-000006SixC1	YES	NO
L_000003Three-000004FourC1	NO	NO
L_000004Four-000005FiveC1	NO	NO
L_000007Seven-000005FiveC1	YES	NO
L_000006Six-000007SevenC1	YES	NO

Plot title

The page is broken into two basic sections: the left and right half. The right half of the display allows selection of the scenarios that should be included on a given plot. The left half allows selection of the quantities that will be charted on the plot.

Selecting Scenarios to Plot

The right half of the display gives a table of all scenarios that were studied. Change the **Plot?** field to *YES* to include a scenario in the current plot. The Plot? field is a toggleable field and can be changed from *YES* to *NO* and back again by double-clicking on the appropriate entry. Any scenario whose Plot? field is set to *YES* will be included on the same plot once the **Plot** button is clicked.

The **Always include base case** box should be checked if the base case scenario should be included in the plot along with any other scenarios selected. This option is not available for QV results plotting.

Before a plot can be created, the quantities to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Selecting Quantities to Plot

The left half of the display gives the various options for selecting what quantities should be plotted.

The **Horizontal axis value** drop-down contains any parameters that were monitored. Additionally, for the PV analysis, this drop-down also contains the independent quantities (Nominal Shift, Export, and Import) recorded during the analysis. If the **Pre-contingency** box is checked, the x values for each plotted scenario will come from the base case instead of the results of a given scenario. The Pre-contingency check box is not available for QV results plotting.

The **Vertical axis value type** drop-down contains a list of all the types of values that were monitored. The type selected in this drop-down will determine the actual elements that are listed in the **Plot values for these elements** table. To include an element in the plot, change the **Plot?** field to *YES* for that element. The Plot? field is a toggleable field and can be changed from *YES* to

NO and back again by double-clicking on the appropriate entry. Any element whose Plot? field is set to *YES* will be included in the same plot once the **Plot** button is clicked.

Before a plot can be created, the scenarios to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Plot Title

Optional field that can be set to add a title to a plot and distinguish one plot from another.

Plot

After selecting the scenarios and values to plot, click this button to actually generate the plot. The plot will appear in its own window. The number of plots that can be created is limited only by computer system memory.

Right-clicking on a plot will expose a local menu with several options. The plot can be saved as a bitmap, Windows metafile, JPEG, or text file by clicking **Save** and selecting the appropriate file type. **Copy** will allow the plot to be copied and pasted into other applications. **Print** will send the plot to a printer that can be configured by selecting the **Printer Setup** option. Selecting **Close** will close the plot window.

Clear

Click this button to reset all of the **Plot?** fields to *NO* for both the scenarios to plot and the values to plot.

PV/QV Curves Results: Track Limits

The Track Limits sub-tab is found on the [PV Results](#) tab of the [PV Curves dialog](#) or the [QV Results](#) tab of the [QV Curves dialog](#). The results displayed are similar for both the PV and QV analysis. This page provides access to the results of [tracking devices at limits](#) which are determined in the same manner for both the PV and QV analysis.

Overview Plot **Track Limits**

Filter out devices that never hit or backoff a limit during the PV run

Generator Var Limits Shunt Var Limits LTC Var Limits Line Thermal Limits Interface Thermal Limits

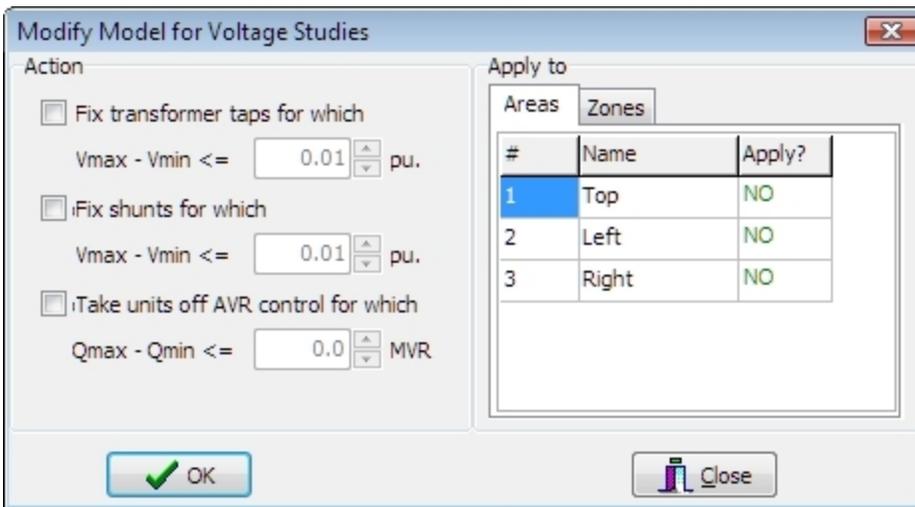
Records Set Columns Options

	Number of Bus	Name of Bus	ID	0.00	100.00	200.00	225.00	250.00	300.00
1	1	One	1	Within Range	At Min	At Min	Within Range	Within Range	Within Range
2	2	Two	1	Within Range	Within Range	At Max	At Max	At Max	At Max
3	4	Four	1	Within Range					
4	6	Six	1	Within Range	At Max				
5	7	Seven	1	Within Range	Within Range	At Max	At Max	At Max	At Max

The page is divided into several additional sub-tabs for the different types of devices that can be tracked. Tables on each tab are similar in that they list all of the devices of that type that were tracked. For PV analysis, each column of the table is the transfer amount at which the device limits were checked. For QV analysis, each column of the table is the per-unit voltage set-point for the study bus during the scenario in which the limits were checked. The results show where each element was located relative to its min/max limits during each step of the analysis using the indicators of *Within Limits*, *Within Range*, *At Max*, and *At Min*. To simplify the display and exclude those elements that were always within their limits during all steps of the process, check the box **Filter out devices that never hit or backoff a limit during the PV run**.

PV/QV Refine Model

Modeling idiosyncrasies may exist that cause premature loss of convergence during either a PV or QV analysis. The PVQV add-on has a tool to modify the currently open case in an attempt to fix some of these idiosyncrasies. This is the Refine Model tool that is available on the [Add Ons](#) ribbon tab in the **PV and QV Curves (PVQV)** ribbon group.



The case can be refined with the following options:

Fix transformer taps

If there are transformers that have Vmax and Vmin that are very close together, the power flow may have a difficult time converging. This option allows the user to fix all transformer taps at their present values if their Vmax - Vmin is less than or equal to the user specified tolerance.

Fix shunts

If there are switched shunts that have Vmax and Vmin that are very close together, the power flow may have a difficult time converging. This option allows the user to fix all switched shunts at their present values if their Vmax-Vmin is less than or equal to the user specified tolerance.

Take units off AVR control

If there are generators that have Qmax and Qmin that are very close together, the power flow may have a difficult time converging. This option allows the user to remove these units from AVR control, thus locking their MVAR output at its present value, if their Qmax - Qmin is less than or equal to the user specified tolerance.

Apply to

These refinements will only be applied to those areas or zones that have the **Apply?** field set as *YES* in this dialog box. This field can be toggled between *YES* and *NO* by left-clicking on the appropriate entry.

OK

Click this button to make the selected changes to the currently open power flow case. After the changes are made the **Close** button must be clicked to close the dialog. If any changes have been made and the dialog is closed, the user will be prompted to save the case to a file.

Close

Click this button to close the dialog without implementing any changes to the power flow case in memory.

PowerWorld Simulator Optimal Power Flow Overview

Note: The OPF option in PowerWorld Simulator is only available if you have purchased the OPF add-on to the base package. To learn more about the OPF, please feel free to read through the information contained in these help files. [Contact PowerWorld Corporation](#) for details about ordering the OPF version of Simulator.

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the standard mode Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. However with the optimal power flow (OPF) enhancement, Simulator OPF can also solve these equations using an OPF. In particular, Simulator OPF uses a linear programming (LP) OPF implementation.

All of the OPF commands and options are accessed using the LP OPF main menu item. Other commands in this menu are used to specify input options, see results, and store/retrieve OPF specific data into auxiliary files.

The purpose of an OPF is to minimize an [objective \(or cost\) function](#) by changing different [system controls](#) taking into account both [equality and inequality constraints](#) which are used to model the power balance constraints and various operating limits.

In Simulator OPF the LP OPF determines the optimal solution by iterating between solving a standard power and then solving a linear program to change the system controls to remove any limit violations. See [OPF Primal LP](#) for more details.

OPF Objective Function

The objective of the OPF algorithm is to minimize the OPF objective function, subject to various equality and inequality constraints. Since the objective of the OPF is to minimize an objective function, what objective function is used has a significant impact on the final solution.

Currently two objective functions are available in Simulator OPF: Minimum Cost and Minimum Control Change. Minimum Cost attempts to minimize the sum of the total generation costs in specified areas or super areas. Minimum Control Change attempts to minimize the sum of the absolute values of the changes in the generation in the specified areas or super areas.

To include an area or super area in the OPF objective function, simply change the Area AGC Status field to "OPF" on the [OPF Area Records Display](#) or the Super Area AGC Status field to "OPF" on [the OPF Super Area Records Display](#). This gives you great flexibility in defining the OPF study. For example you can set the OPF to minimize costs for the entire system, or just selected areas or super areas.

OPF Equality and Inequality Constraints

In solving a constrained optimization problem, such as the OPF, there are two general classes of constraints, equality and inequality. Equality constraints are constraints that always have to be enforced. That is, they are always "binding". For example in the OPF the real and reactive power balance equations at system buses must always be satisfied (at least to within a user specified tolerance); likewise the area MW interchange constraints. In contrast, inequality constraints may or may not be binding. For example, a line MVA flow may or may not be at its limit, or a generator real power output may or may not be at its maximum limit.

An important point to note is because the OPF is solved by iterating between a power flow solution and an LP solution, some of the constraints are enforced during the power flow solution and some constraints are enforced during the LP solution. The constraints enforced during the power flow are, for the most part, the constraints that are enforced during any power flow solution. These include the bus power balance equations, the generator voltage set point constraints, and the reactive power limits on the generators. What differentiate the LP OPF from a standard power flow are the constraints that are explicitly enforced by the LP. These include the following constraints:

[Equality Constraints](#)

[Inequality Constraints](#)

OPF Equality Constraints

Area MW Interchange

The area MW interchange constraints are enforced during the LP for those areas that have an **AGC Status** equal to "OPF" provided the area is not part of a super area that is also set on AGC. The **AGC Status** field for an area can be set using the [OPF Area Records display](#), while the **AGC Status** field for the super area (if any) is set using the [OPF Super Area Records display](#). Areas whose interchange is enforced during the LP do not have their interchange enforced during the power flow solution; during the power flow these areas are treated as though they were off of AGC (and hence the output of generators in that area is not varied during the power flow).

It is perfectly acceptable to have some areas on "OPF" AGC control and to have other areas on the more traditional power flow area AGC such as "ED" or "Part. AGC." The interchange for such areas is controlled during the power flow solution.

Following a successful solution, marginal costs are calculated for the area interchange constraints; these values are displayed on the [OPF Area Records display](#) and can be contoured. See [OPF Marginal Costs](#) for details.

Bus MW and Mvar power balance

Enforced during the power flow solution. Following a successful solution, marginal costs are calculated for the bus MW (real power) balance constraint; these values are displayed on the [OPF Bus Records display](#) and can be contoured.

Generator Voltage Setpoint

Enforced during the power flow solution. Following a successful solution, marginal costs are calculated for the voltage setpoint constraint; these values are displayed on the [OPF Bus Records display](#).

Super Area MW Interchange

Super area interchange constraints are enforced similar to the area constraints. That is, super area interchange constraints are enforced during the LP only for those super areas that have an **AGC Status** equal to "OPF." The **AGC Status** field can be set using [the OPF Super Area Records display](#). During the power flow solution such super areas are treated as though they were off of AGC.

Interface MW limits when treated as Equality

Interface MW limits are enforced during the LP solution. Interface MW limits are normally treated as inequality constraints (see [Inequality Constraints](#)), however they can optionally be treated as equality constraints. See the [Interface Dialog](#) for information on how to treat the limit as an equality constraint.

Transmission Line and Transformer (Branch) MVA limits

Branch MVA limits are enforced during the LP solution. Branch MVA limits are normally treated as inequality constraints (see [Inequality Constraints](#)), however they can optionally be treated as equality constraints. See the [Line Transformer Dialog](#) for information on how to treat the limit as an equality constraint.

OPF Inequality Constraints

The following classes of inequality constraints are enforced during the OPF solution.

Generator real power limits

Generator real power limits are enforced during the LP solution.

Generator reactive power limits

Generator reactive power limits are enforced during the power flow solution.

Interface MW limits

Interface MW limits are enforced during the LP solution. In short, interface records are used to represent the aggregate flow through a number of different devices (see [Interface Records](#) for details). During the LP the MW flow through the interface is constrained to be less than or equal to a user specified percentage of its limit, provided the interface is active for enforcement. For an interface to be active for enforcement the following three conditions must be met:

- Interface enforcement must not be disabled for the case. This field can be set from either the [OPF Options and Results dialog](#) or the [OPF Interface Records display](#). The default is that case interface enforcement is not disabled. Also note that interface flow is limited to a percent of its limit as specified by the interface's [Limit Monitoring Settings](#).
- Interface enforcement must be active for at least one of the interface's areas. Note, an interface is assumed to be in each area that contains at least one of its components. This field can be set from [the OPF Area Records display](#). Note: the default is that interface enforcement is **not active**, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual interface. This field can be set from [the OPF Interface Records display](#) or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each interface that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcement, which is shown on the [OPF Interface Records display](#).

Transmission Line and Transformer (Branch) MVA Limits

Transmission line and transformer (branch) MVA limits are enforced during the LP solution. During the LP the branch line flow is constrained to be less than or equal to a user specified percentage of its limit, provided the branch is active for enforcement. For a branch to be active for enforcement the following three conditions must be met:

- Line/Transformer enforcement must not be disabled for the case. This field can be set from either the [OPF and Results Options dialog](#) or the [OPF Line/Transformer Records display](#). The default is that case line/transformer enforcement is not disabled. Also note that the branch flow is limited to a percent of its limit as specified by the branch's [Limit Monitoring Settings](#).
- Branch enforcement must be active for the branch's area. For tie-lines enforcement must be active for either area. This field can be set from the [OPF Line/Transformer Records display](#). The default is that branch enforcement is **not active**, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual branch. This field can be set from the [OPF Line/Transformer Records display](#) or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each branch that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcement, which is shown on the [OPF Line/Transformer Records display](#).

Determining Set of Active Inequality Constraints

A key issue in quickly solving the OPF is for the LP to effectively determine the set of active inequality constraints. Currently this includes the line MVA limits and the interface MW limits. Because the speed of the LP varies as the cube of the number of constraints active in the LP basis, it is extremely important to keep this number as small as possible. Therefore it would be very computationally prohibitive to setup an inequality constraint for each transmission line and interface (except in very small systems.)

The solution of setting up constraints **only** for those inequality constraints that are actually violating their limits is a step in the right direction, but suffers from the problem that during a solution a line may initially be violating its limit and then after the first iteration it is no longer violating. However if it is not subsequently included as a constraint during the next iteration the solution may simply oscillate between enforcing/unenforcing this constraint. This problem can be resolved by keeping that constraint in the basis even though it is no longer binding.

However this raises a question about how to handle these constraints during future OPF solutions. For example what would happen if a user solved the OPF, and then immediately resolved the OPF. Following the first solution the constraint would be enforced so that it may actually be less than its limit. However if this constraint is not included in the LP basis during the next solution the constraint may immediately violate during the first iteration, requiring a number of iterations just to return to the original initial solution.

Simulator solves this issue by keeping track of the enforced constraints from one solution to the next. Constraints are only removed from the basis if they fall below a specified percentage of their limit. This percentage is enterable on the [Constraint Options page](#) of the [OPF Options and Results Dialog](#). This prevents the set of constraints in the basis from building up over time as a number of different system conditions (and hence constraints) are studied. Also, at any time this set of constraints can be cleared using the **Initialize OPF Button** on the [OPF Options and Results Dialog](#).

Also, the user is free to specify that a particular constraint **always** be included in the basis. This is done by toggling the **Constraint** field to "Always" on the [OPF Line/Transformer Records](#) or [OPF Interface Records](#) displays.

OPF Unenforceable Constraints

The goal of the LP OPF is to minimize the objective function subject to the user specified constraints. However there is no guarantee that it is even possible to simultaneously satisfy all of the specified constraints. In fact, it is quite easy to create a system in which all of the constraints **cannot** be enforced. A simple example is a two bus system consisting of a single generator supplying a single load through a transmission line. If the transmission line MVA rating is below the MVA of the load then it is impossible to supply this load while simultaneously satisfying the transmission line constraint. In Simulator OPF such a situation is known as an unenforceable constraint. In studying large systems, such as the U.S FERC 715 cases, such situations actually appear to be quite common. Seemingly unenforceable constraints are often due to a lack of controls available to the LP OPF or due to faulty limits entered in the case. In such cases unenforceable constraints can be corrected by making more controls available to the LP OPF or correcting the limits.

Simulator OPF allows you to solve systems with unenforceable constraints by only enforcing those constraints that have a marginal cost below a user specified tolerance. These tolerances are specified on the [OPF Options Constraint Options Page](#). Any constraints that have marginal costs above these values are not enforced, including any unenforceable constraints. This functionality is implemented in Simulator OPF through the use of slack variables. Slack variables are artificial variables introduced during the LP solution in order to satisfy the constraints with the slack variable costs equal to the user specified values. Then, during the LP solution the slack variables are usually removed from the LP basis. The only time this does not occur is if the constraint can not be enforced with a marginal cost less than the specified value. The number of unenforceable constraints are shown on the [OPF Option Solution Results Page](#).

OPF Marginal Costs

During any constrained minimization there is a cost associated with enforcing the equality constraints and the binding inequality constraints. These costs are known as the marginal costs.

In Simulator OPF marginal costs are calculated for the following record types:

Bus MW Equality Constraints

The Bus MW marginal costs tell the incremental cost to supply one additional MW of load at the specified bus. These values can be viewed on the [OPF Bus Records display](#); they can also be contoured or viewed on the one-lines using [bus fields](#).

In the absence of any binding inequality constraints (such as Line MVA constraints) all of the bus marginal costs in an area should be identical. Bus marginal costs can only be determined for buses that are in areas or super areas on OPF control.

Area MW Equality Constraints

The Area MW marginal costs tell the incremental cost for the specified area to import one additional MW of load **from the system slack bus**. These values can be viewed on [the OPF Area Records display](#); they can also be contoured or viewed on the one-lines [using area fields](#). In the absence of any binding inequality constraints the area MW marginal cost is identical to the bus MW marginal costs for all the buses in the area. When there are binding inequality constraints this is no longer the case.

Super Area MW Equality Constraints

The Super Area MW marginal costs are identical to the area marginal costs except they apply to super areas rather than areas.

Interface MW Constraints

The Interface MW marginal costs tell the incremental cost of enforcing the interface MW constraints. These values are only nonzero if the interface constraint is actually active (binding); they can be viewed using the [OPF Interface Records display](#).

Line/Transformer MVA Constraints

The Line/Transformer marginal costs tell the incremental cost of enforcing the line or transformer MVA constraint. These values are only nonzero if the line or transformer constraint is actually active; they can be viewed using the [OPF Line/Transformer Records display](#).

OPF Primal LP

Go to the [Add Ons](#) ribbon tab and select **Primal LP** from the **OPF** ribbon group to solve the OPF using the primal LP algorithm.

In Simulator OPF the LP OPF determines the optimal solution by iterating between solving a standard power flow and then solving a linear program to change the system controls to remove any limit violations. The basic steps in the LP OPF algorithm are

1. Solve the power flow
2. Linearize the power system about the current power flow solution. Both constraints and controls are linearized.
3. Solve the linearly-constrained OPF problem using a primal LP algorithm, computing the incremental change in the control variables. Slack variables are introduced to make the problem initially feasible. That is, the slack variables are used to satisfy the equality and inequality constraints. The slack variables typically have high costs so that during the iteration the slack variables change to satisfy the constraints. The LP then determines the optimal, feasible solution for the linear problem.
4. Update the control variables and resolve the power flow.
5. If the changes in the control variables are below a tolerance then the solution has been reached; otherwise go to step 2.
6. Finish by resolving the power flow.

OPF Future Enhancements

While we certainly plan on introducing additional functionality in future releases, we do want to be as clear as possible about what functionality is not currently provided.

The current version of Simulator OPF allows users to calculate the optimal solution to a power system using generator real power MW outputs and phase shifters as controls, while enforcing area, super area, interface MW and line/transformer MVA constraints. Marginal losses can also be included in the OPF calculation.

Some functionality that is **not** included in the current version of Simulator OPF, and which we hope to include in future versions, include the following:

- Enforcing bus low/high voltage magnitudes as limits
- Including additional devices as controls, such as generator voltage setpoints, LTC transformers, switched shunts.
- Allowing the optimization of different cost functions.
- Additional functionality as suggested by customers.

OPF Options and Results

The OPF Options and Results dialog allows you to customize the OPF solution. To display this dialog, go to the [Add Ons](#) ribbon tab and select **Options and Results** from the **OPF** ribbon group. The dialog consists of three general pages; Options, Results and LP Solution Details.

The Options page has four tabs as well, [Common Options](#), [Constraint Options](#), [Control Options](#), and [Advanced Options](#)

The Results page has four tabs, [Bus MW Marginal Price Details](#), [Bus Mvar Marginal Price Details](#), [Bus Marginal Controls](#), and [Solution Summary](#).

The LP Solutions Page has five tabs, [All LP Variables](#), [LP Basic Variables](#), [LP Basis Matrix](#), [Inverse of LP Basis](#), and Trace Solution.

The dialog also has several buttons at the bottom of the display:

OK, Cancel

Select to close the dialog. Selecting **OK** saves your changes while **Cancel** does not. Note that changes are also saved anytime you select **Solve LP OPF** or **Single Outer Loop**.

Solve LP OPF

Solves the OPF using the Primal LP algorithm. Equivalent to selecting [Primal LP](#) from the **Optimal Power Flow** ribbon group on the [Add Ons](#) ribbon tab.

Single Outer Loop

Does a single outer loop of the Primal LP algorithm.

Initialize LP OPF

Returns the LP OPF variables to their original states.

Print

Prints the selected page of the dialog.

Help

Displays this help page. To view help for a particular page place the cursor on the page and press the F1 key.

OPF Options: Common Options

The [OPF Dialog](#), Common Options page displays general options associated with the OPF solution. The display contains the following fields:

Objective Function

Allows a choice of solving the LP using either a minimum cost or a minimum control change objective function.

OPF Reserves Options

These options are only available with the [OPF Reserves add-on](#).

Include OPF Reserve Requirements

Check this box to include OPF reserve constraints in the OPF calculation.

Use a Single Control for Up/Down Regulation

Check this box to use a single bidirectional control for regulating reserve up or down. A unit will provide the same amount of regulating reserve up or down.

LP Control Variables**Disable All Phase Shifter Controls**

Prevents phase shifters from attempting to control devices during the OPF solution.

Disable All Generator MW Controls

Prevents generators from shifting MW output during the OPF solution.

Disable All Load MW Controls

Prevents loads from shifting MW demand during the OPF solution.

Disable Area-toArea MW Transaction Controls

Prevents MW transactions between areas from being dispatched during the OPF solution.

Disable DC Transmission Line MW Controls

Prevents shifting of DC Transmission Line MW transfers during the OPF solution.

LP Options**Maximum Number of LP Iterations**

Maximum number of allowable iterations for the LP portion of the LP OPF. How many iterations are required to obtain a solution depends, among other things, upon the number of breakpoints in the control cost models. Since each LP iteration can only move from one breakpoint to the next, the finer the model the more iterations required. However the LP is quite fast so a large number of iterations can be performed quite quickly. Default = 1000.

Phase Shifter Cost (\$ / Degree)

Specifies the assumed cost for moving phase shifting transformer taps away from their initial values. The purpose for this fictitious cost is approximate the cost of actually changing the angle of a phase

shifting transformer, and to avoid large changes in phase shifter angles that have very little impact on the system. This field may be zero. Default = \$ 0.10 / Degree.

Calculate Bus Marginal Cost of Reactive Power

When this option is checked, the OPF algorithm will also calculate the marginal cost of reactive power at each bus. Typically the result of interest from the OPF algorithm is the MW marginal cost of each bus (the LMP), but the MVAR marginal cost can be determined as well.

Save Full OPF Results in PWB File

When checked, Simulator will store the full set of results, including the LP matrix, in the PowerWorld Binary case file.

Do Detailed LP Logging

When checked, Simulator will write details on the LP algorithm solution during each pivot of the LP matrix. This is useful for debugging LP solution issues when running a LP OPF solution.

Start with Last Valid OPF Solution

When checked, Simulator will start the LP OPF solution process using the most recent OPF solution as the initial conditions.

OPF Options: Constraint Options

The OPF Dialog, Constraint Options page displays options associated with the enforcement of the constraints by the OPF. The display contains the following fields:

Line/Transformer Constraints

Disable Line/Transformer MVA Limit Enforcement

Select to disable enforcement of Line/Transformer MVA constraints for the entire case.

Percent Correction Tolerance

Specifies a tolerance for the enforcement of line/transformer MVA flows. The tolerance is necessary to prevent solution oscillations due to the non-linear nature of the actual constraints.

Violated elements are always enforced to their limits multiplied by the MVA Enforcement Percentage. If power systems were completely linear then following the LP solution the constraint would actually be equal to this value. However because of nonlinearities, the constraint is close to this value but usually not identical to the value. The Percent Correction Tolerance is used to tell the OPF how close is close enough. Provided all the constraints are violating their limits by less than the correction tolerance percentage the optimal solution is assumed to have been found. You may set this value as low as you like, but setting it too close to zero may result in convergence difficulties. The default is 2 percent.

MVA Auto Release Percentage

Specifies a MVA level at which transmission lines can be released as an OPF constraint equation if the branch MVA flow falls below the level specified.

Maximum Violation Cost (\$/MWhr)

If a branch MVA limit cannot be enforced during an OPF solution, the branch will be assigned a fictitious cost of enforcement equal to this value. This value is usually rather large in order to easily determine where the unenforceable constraint is occurring. The default value is 1000 \$/MWhr.

Enforce Line/Transformer MW Flow Limits (Not MVA)

Checking this box will cause Simulator to treat the limits of the transmission elements as MW limits instead of MVA limits. Thus Simulator will report violations on these elements in the OPF based on the MW flow of the element versus the element's MVA rating.

Interface Constraints

Disable Interface MW Limit Enforcement

Select to disable enforcement of Interface MW constraints for the entire case.

Percent Correction Tolerance, MW Auto Release Percentage, Maximum Violation Cost (\$/MWhr)

These fields are equivalent to the entries described above for Line/Transformer MVA Constraints except that they apply to Interface MW constraints.

Monitor/Enforce Contingent Interface Limits

This option allows you to choose if contingency elements in interfaces should be enforced during the OPF solution. Even if they are not enforced, the flows on the remaining elements in the interface will be monitored. The choices you have for enforcing contingency elements in interfaces during the OPF are Never, Power Flow/OPF but not CA/SCOPF, or All Applications including CA/SCOPF. The reason

CA ([contingency analysis](#)) and SCOPF ([Security Constrained OPF](#)) are singled out is because those two tools are already looking at contingency actions separately, outside of the interface definitions. Therefore it may be desired to ignore the inclusion of contingency elements within an interface definition when using these two tools.

Phase Shifting Transformer Regulation Limits

Disable Phase Shifter Regulation Limit Enforcement

Select to disable enforcement of Phase Shifter regulation limit constraints for the entire case.

For a phase shifter, there are two limits which can be enforced during the OPF—the MVA branch rating, and the regulation minimum and maximum MW flow. Checking this option will globally disable the enforcement of the regulation minimum and maximum MW flow for all phase shifters.

In Range Cost (\$/degreehr)

The cost of changing a phase shifter angle away from zero degrees in \$/degreehr, when the angle is within the allowable range of the phase shifter.

Maximum Violation Cost (\$/MWhr)

If a phase shifter regulation limit cannot be enforced during an OPF solution, it will be assigned a fictitious cost of enforcement equal to this value. This value is usually rather large in order to easily determine where the unenforceable constraint is occurring. The default value is 1000 \$/MWhr.

Limit Monitoring Settings...

This button opens the [Limit Monitoring Settings](#) dialog, which allows you to change the enforcement percentages for monitored elements.

OPF Options: Control Options

The LP OPF Dialog, Control Options page displays options for generator control and power flow solution. The display contains the following options:

Generator Control Options

Allow Commitment of Fast Start Generators

If this option is checked, then generators designated as Fast Start generators can be turned on or "committed" if the OPF routine determines that doing so would reduce the overall generating costs of the system.

See the Fast Start description in the help on [OPF Generator Records](#) for a more detailed description of the Fast Start option of generators.

Allow Decommitment of Fast Start Generators

If this option is checked, then generators designated as Fast Start generators can be turned off or "de-committed" if the OPF routine determines that doing so would reduce the overall generating costs of the system.

See the Fast Start description in the help on [OPF Generator Records](#) for a more detailed description of the Fast Start option of generators.

Modeling Generators without Piecewise Linear Cost Curves

The following fields specify how the OPF should handle generators that are specified as having a cubic cost model. Because the OPF is based upon an LP implementation, all control costs must be modeled using piecewise linear cost curves. These options permit an automatic conversion of cubic models to piecewise linear models. Alternatively, you can very easily convert the cubic models manually using the **# Cost Curve Points** field on [the OPF Generator Records display](#) or using the [generator dialog](#).

Generators Cost Models

This field specifies how generators with cubic cost models should be handled in the OPF. The field has three values

Ignore Them - Generators with cubic cost models are Ignored during the OPF solution. That is, they are considered as though their AGC status was off.

Change to Specified Points per Curve - A piecewise linear cost model is automatically inserted for the generator with a fixed number of points specified in the **Total Points Per Cost Curve** field described below. This curve will approximate the generator's cubic cost model as closely as possible; the existing cubic model is not modified. This is the default value.

Change to Specified MWs per Segment - A piecewise linear cost model is automatically inserted for the generator such that each segment in the cost model covers the amount of MWs specified in the **MWs per Cost Curve Segment** field described below. This curve will approximate the generator's cubic cost model as closely as possible; the existing cubic model is not modified.

Total Points Per Cost Curve

Specifies the total number of segments that should be automatically inserted into the piecewise linear cost models for those generators that are modeled using cubic cost functions. This is only done if the **Generator Cost Modeling** field is **Change to Specified Points per Curve**. Default = 5.

MWs per Cost Curve Segment

Specifies the number of MWs for each segment of the piecewise linear cost models that are automatically inserted for those generators that are modeled using cubic cost functions. This is only done if the **Generator Cost Modeling** field is **Change to Specified MWs per Segment**. Default = 10 MW.

Save Existing Piecewise Linear Cost Curves

Generators that are modeled with cubic cost curves may have existing piecewise linear cost curves which may have been manually entered by the user. These curves may or may not resemble the cubic cost function. During the OPF solution the existing piecewise linear cost curves are replaced with the auto-created cost curves. If this option is checked then the existing piecewise linear cost curves are restored at the end of the OPF. The default and recommended option is false since this allows one to view the actual cost curves used by the OPF.

If you would like to use a particular piecewise linear cost function simply make sure that the generator is modeled using the piecewise linear model, which can be set on the [OPF Generator Records](#) display.

Modeling of OPF Areas/Superareas

During the Initial OPF Power Flow Solution

Choose what manner of generation control you wish to be employed in the FIRST power flow solution the OPF will perform, which will establish the base case load flow condition for performing the subsequent OPF generation dispatch.

During Stand-Alone Power Flow Solutions

Choose what manner of generation control you wish to be employed in all load flow solutions FOLLOWING the initial load flow solution. In other words, after the LP OPF routine has determined the new generation dispatch, what type generation dispatch should be used during the normal load flow solution.

NOTE: it is NOT recommended that you use Economic Dispatch in this case, although it is an available option. The reason it is not recommended is that you will remove the optimal dispatch (including constraints) just determined by the OPF in favor of lowest cost economic dispatch, which will likely result in the re-introduction of overloaded elements that were corrected by the OPF dispatch in the first place.

Treat Area/Superarea MW Constraints as unenforceable even when the ACE is less than the AGC Tolerance (default is checked)

This option by default is checked, which means that if Simulator solves an OPF and the generators in an area or super area are all either at their minimum or maximum outputs, and the area control error is still not 0, then the area MW constraint is considered unenforceable and Simulator will assign a large penalty cost (usually \$5,000/MWhr) to the LMP's of the area. However, if you deselect this option, you give Simulator the additional control of checking the ACE mismatch against the AGC tolerance for the area, and if the ACE mismatch is less than the AGC tolerance, Simulator will "acquire" the mismatch amount from the system slack bus for the area, and the area will not be considered unenforceable in the OPF solution. The AGC tolerance is usually small, and therefore the amount of power taken from the system slack bus will also be small. The purpose of this option is to allow for valid LMP solutions in an area that is meant to reach its full minimum or maximum generator capability, but that losses and numerical rounding results in the ACE mismatch deviating slightly from 0. It is not advised to uncheck this option if you have areas with large AGC tolerances.

OPF Options: Advanced Options

Detection of LP Cycling

This option allows for advanced tweaking of when cycling is detected in the LP. It is recommended that users leave the default settings in place.

Minimum Number of Degenerate Iterations to Assume Cycling

To prevent inaccurate identification of cycling in the LP solution, the minimum number of degenerate iterations before cycling is identified can be specified manually in this box.

Minimum Number of Degenerate Iterations Multiplied by Number of Tableau Rows

To prevent inaccurate identification of cycling in the LP solution, the minimum number of degenerate iterations as a multiple of the number of tableau rows can be specified manually in this box.

Number of Sequential Degenerate Iterations for Last Case

This displays the number of degenerate iterations detected when performing the most recent OPF solution.

Power Flow Recalculation

Choose one of the options to determine how often the power flow is resolved. The three options are, "When total generator MW change > than tolerance"; "After each LP solution"; and "Only at end of LP OPF". If the power flow is not resolved, linear sensitivities are used to determine which constraints enter the LP tableau.

Total Generator Change Tolerance (MW)

Specifies the total generator change tolerance. The default is 500 MW.

After each LP solution

Resolves the power flow upon completion of each LP solution

Only at end of LP OPF

Resolves the power flow only after the LP OPF has completed.

OPF Options: Solution Summary

The OPF Dialog, Solution Summary page displays general results from the last OPF solution. Click the **Save as Aux** button to save these results to a Simulator Auxiliary file.

The display contains the following fields, none of which can be directly changed:

General Results**Solution Start Time, Solution End Time**

The starting time and ending time of the OPF solution algorithm.

Total Solution Time, Last Solution Status

Time and status of the last OPF solution.

Number of LP Iterations

Total number of LP iterations used during the last OPF solution. The maximum number of iterations is specified in the [Maximum Number of Iterations](#) field of the General Options page.

Initial Cost Function Value

Initial value of the [OPF cost function](#). During the OPF the solution algorithm seeks to minimize the cost function, subject to the equality and inequality constraints.

Final Cost Function Value

Final value of the OPF cost function.

Final Slack Cost Value

The slack cost value is an artificial cost that is only non-zero when there are one or more unenforceable constraints.

Final Total Cost Value

The addition of the final cost function value and the final slack cost value.

Number of Buses in OPF

This field contains the total number of buses that are in areas or super areas that are on OPF control. Thus this field need not be equal to the total number of buses in the case. Marginal costs are only calculated for buses in OPF controlled areas or super areas.

Highest Bus Marginal Cost, Lowest Bus Marginal Cost, Average Bus Marginal Cost

Highest, lowest and average marginal cost for the buses that are in OPF controlled areas or super areas.

Bus MC Standard Deviation

The standard deviation of the Bus Marginal Cost.

Area and Superarea Constraints**Unenforceable Area Constraints**

Number of unenforceable area constraints.

Unenforceable Superarea Constraints

Number of unenforceable superarea constraints.

Line MVA Constraints

The **Line MVA Constraints** fields present results associated with the enforcement of the line MVA constraints.

Number of Initial Violations, MVA Sum of Initial Violations

Total number of lines that initially exceeded their MVA limits and were eligible for enforcement by the OPF. For these lines only, the **MVA Sum of Initial Violations** field contains sum of the absolute values of the line's actual MVA flow minus the line's MVA limit.

Number of Binding Lines

Total number of lines that are constrained to their limit value.

Highest Line MVA Marginal Cost

The highest Marginal Cost for an MVA change on a line.

Number of Unenforceable Violations

Total number of lines whose MVA flows can not be enforced by the OPF using the available controls.

MVA Sum of Unenforceable Violations

For all the unenforceable lines, this field contains the sum of the absolute values of the line's actual MVA flow minus the line's MVA limit.

Interface MW Constraints

The **Interface MW Constraints** field present results associated with the enforcement of the interface MW constraints.

Number of Initial Violations, MW Sum of Initial Violations

Total number of interfaces that initially exceeded their MW limits and were eligible for enforcement by the OPF. For these interfaces only, the **MW Sum of Initial Violations** field contains the sum of the absolute values of the interface's actual MW flow minus the interface's MW limit.

Number of Binding Interfaces

Total number of interfaces that are constrained to their limit value.

Highest Interface MW Marginal Cost

The highest Marginal Cost for an MVA change on an interface.

Number of Unenforceable Violations

Total number of interfaces whose MW flows can not be enforced by the OPF using the available controls.

MW Sum of Unenforceable Violations

For all the unenforceable interfaces, this field contains the sum of the absolute values of the interface's actual MW flow minus the interface's MW limit.

Generator MW Control Limit Violations

The **Generator MW Control Limit Violations** fields present results associated with the enforcement of the generator MW limit constraints.

Number of Initial Violations, MW Sum of Initial Violations

Total number of generators that initially exceeded their MW limits and were eligible for enforcement by the OPF. For these generators only, the **MW Sum of Initial Violations** field contains the sum of the absolute values of the generators' actual MW output minus the generators' MW limits.

Number of Unenforceable Violations

Total number of generators whose MW limits can not be enforced by the OPF using the available controls.

MW Sum of Unenforceable Violations

For all the generators with unenforceable limits, this field contains the sum of the absolute values of the generators' actual MW output minus the generators MW limits.

Transformer Regulation Constraints

The **Transformer Regulation Constraints** fields present results associated with the enforcement of phase shifting transformer regulation.

Number of Initial Violations

Total number of phase shifting transformers that initially exceeded their MW set point limits and were eligible for enforcement by the OPF.

Number of Binding Constraints

Total number of phase shifting transformers that are set at their minimum or maximum regulation value.

Number of Unenforceable Violations

Total number of phase shifting transformers whose MW regulation limits can not be enforced by the OPF using the available controls.

Fast Start Generators

The **Fast Start Generators** fields present results associated with the commitment/decommitment of fast start generators.

Number of Generators Turned On

Total number of fast start generators turned on during the OPF solution

Number of Generators Turned Off

Total number of fast start generators turned off during the OPF solution

OPF Options: All LP Variables

The OPF Dialog, All LP Basic Variables page displays the basic and non-basic variables associated with the final LP solution. This page is usually only of interest to users interested in the specifics of the LP solution. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the LP variables, showing the following fields for each:

ID

Variable identifier.

Original Value

The initial value of the LP variable before OPF optimization.

Value

The final value of the LP variable after OPF optimization.

Delta Value

The difference between the original value field and the value field.

BasicVar

Shows the index of the basic variables in the LP basis. If the value is zero, the variable is non-basic.

NonBasicVar

Shows the index of the non-basic variable. If the value is zero, the variable is basic.

Cost(Down)

The cost associated with decreasing the LP variable. The field will also show if the variable is at its max or min limit.

Cost(Up)

The cost associated with increasing the LP variable. The field will also show if the variable is at its max or min limit.

Down Range

The available range to decrease the basic variable before a new constraint is hit.

Up Range

The available range to increase the basic variable before a new constraint is hit.

Reduced Cost Up

The cost reduction that would be experienced if an LP variable increases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

Reduced Cost Down

The cost reduction that would be experienced if an LP variable decreases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

At Breakpoint?

Yes, if the LP variable is at a break point.

OPF Options: LP Basic Variables

The OPF Dialog, LP Basic Variables page displays the basic variables associated with the final LP solution. This page is usually only of interest to users interested in the specifics of the LP solution. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the basic variables, showing the following fields for each:

ID

Basic variable identifier.

Original Value

The initial value of the basic LP variable before the OPF optimization.

Value

The final value of the basic LP variable after the OPF optimization.

Delta Value

The difference between the original value field and the value of the basic variable.

Basic Var

Shows the indices of the basic variables in the LP basis.

Cost(Up)

The cost associated with increasing the basic variable.

Down Range

The available range to decrease the basic variable before a new constraint is hit.

Up Range

The available range to increase the basic variable before a new constraint is hit.

OPF Options: LP Basis Matrix

The OPF Dialog, LP Basis Matrix page displays the basis matrix associated with the final LP solution. This page is usually only of interest to users interested in knowing the specifics of the LP solution. Knowing the basis matrix can be helpful in figuring out why a particular power system is exhibiting a particular behavior. The rows of the basis matrix are the binding constraints, while the columns of the basis matrix are the basic variables. The entries in the basis matrix then give the sensitivity of each constraint to each of the basic variables. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, you can right-click anywhere in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

OPF Options: Bus MW Marginal Price Details

The Bus MW Marginal Price Details page displays a grid containing the MW marginal prices computed for an OPF solution. If no OPF solution has been run, the values will all be zero. The grid used for displaying the information is a Case Information Display, which can be modified, sorted, printed, etc., as described in the discussion of [Case Information Displays](#).

One key feature of this display is the breakdown in the bus MW marginal cost into three components: the cost of energy, the cost of congestion, and the cost of losses. Also, while the bus MW marginal cost does not change based on the reference, the costs of energy, congestion, and losses are reference dependent. This reference is set on a per-area (or per-super area) basis in the [Area information dialog OPF tab](#) (or the [Super Area information dialog OPF tab](#)).

OPF Options: Bus MVAR Marginal Price Details

The Bus MVAR Marginal Price Details page displays a grid containing the MVAR marginal prices computed for an OPF solution. If no OPF solution has been run, or if the option to compute MVAR marginal prices has not been selected, the values will all be zero. The grid used for displaying the information is a Case Information Display, which can be modified, sorted, printed, etc., as described in the discussion of [Case Information Displays](#).

OPF Options: Bus Marginal Controls

This display shows the sensitivities of the controls with respect to the cost at each bus. A change in a system control will have the indicated effect in the marginal cost at the system buses. Vice-versa, the marginal cost at a bus is affected by changes in the value of the basic variables.

OPF Options: Inverse of LP Basis

The OPF Dialog, LP Basis Matrix page displays the inverse of the basis matrix. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, right-click somewhere in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

OPF Area Records

Displays OPF specific information about each area record in the case. To show this display select **Optimal Power Flow > Results > Areas** in the [Model Explorer](#). The OPF Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Number, Name

Area's number, between 1 and 999, and its alphanumeric identifier, eight characters maximum.

AGC Status

Area's automatic generation control status. This is the same field shown on the [Area Records Display](#). The field indicates whether or not the area's generation is changing automatically to control the area interchange.

To be included in the OPF, this field MUST be "OPF". The generation costs for areas that are on "OPF" control are included in the [OPF objective function](#); otherwise they are not. Note that if the area is part of a super area that is on AGC control, this field value is ignored.

Double-click on the field to toggle its value.

XF Phase

Specifies whether phase shifting transformers in the area are available as controls. If "Yes" then all transformers in the area which have their Automatic Control Active are available for control; the Automatic Control status for a transformer is set on the [Line/Transformer Dialog](#). If "No" then no transformers in the area are available for control.

Branch MVA

Specifies whether or not the MVA limits should be enforced for transmission lines and transformers that have at least one terminal in this area. For a transmission line or transformer to be included Line/Transformer constraints must not be disabled on the OPF Options and Results Dialog, and the individual line/transformer must be enabled for enforcement on the OPF Line/Transformer MVA Constraints display.

Interface MW

Specifies whether or not the MW limits should be enforced for interfaces that have at least one element in this area. For an interface to be included Interface constraints must not be disabled on the OPF Options and Results Dialog, and the individual interfaces must be enabled for enforcement on the OPF Interface MW Constraints display.

Load MW Dispatch

Specifies whether or not the MW load demand in an area should be included as available for re-dispatch during an OPF solution. In order for loads to be included in OPF re-dispatch, each individual load within the area must be available for control, and have either a fixed cost benefit or a piecewise-linear cost benefit curve provided.

DC Line MW Control

Specifies whether or not DC line MW setpoints can be modified for DC lines that have at least one terminal in this area.

Include Marg. Losses

Specifies whether or not marginal losses should be included for the area during the OPF solution.

MW Marg. Cost Ave

For an OPF solved case this field shows the average of the bus MW marginal costs for all the buses in the area. If there is no congestion then all of the marginal costs should be equal.

MW Marg. Cost St.Dev., Min., Max.

For an OPF solved case these fields show the standard deviation of the bus MW marginal costs for all the buses in the area, the minimum and the maximum bus MW marginal costs.

Report Limits

Specifies whether or not the kV limits should be reported.

Report Min kV, Report Max kV

Specifies the values for minimum and maximum kV levels to report. Defaults are 0 and 9999.

OPF Bus Records

Displays OPF specific information about each bus record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Buses** in the [Model Explorer](#). The OPF Bus Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display. The columns can also be sorted by right-clicking on the heading of the field.

By default the display contains the following fields

Number, Name

Bus's number, between 1 and 2,147,483,647 (equals 2^{31} minus 1), and its alphanumeric identifier.

Area Name

Name of the bus's area.

MW Marg. Cost

Marginal change in the objective function for a one MW change in the real power load at the bus.

MVR Marg. Cost

Marginal change in the objective function for a one Mvar change in the reactive load at the bus.

Volt Marg. Cost

Marginal change in the objective function for a 0.01 per unit change in the voltage setpoint for the bus. This field is only valid at bus's whose terminal voltage is controlled by one or more generators.

Note from the developers – this field is still under construction – do not use it yet.

OPF Generator Records

Displays OPF specific information about each generator record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Generator Records** in the [Model Explorer](#). The OPF Generator Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Number, Name

Number and name of the bus to which the generator is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for this bus. You can also use the local menu to view the [generator's dialog](#).

ID

Alphanumeric ID used to distinguish multiple generators at the same bus.

Area Name of Gen

Name of the generator's area.

AGC

Designates whether the generator's real power output is governed by automatic generation control (AGC). If the AGC field is set to Yes the generator is on AGC in the standard power flow. When a generator is on AGC its real power output is varied automatically, provided the generator is part of an area or super area that is also on automatic control.

In Simulator OPF the default operating mode is that only generators on AGC control are eligible to be OPF controls. In addition, the generator's area or super area must have **AGC Status** of "OPF". However in rare instances you may wish to always make a generator available for control or never make the generator available for control. This value is specified using **OPF MW Control** field.

Fast Start

Designates whether the generator is available as a Fast Start generator during the OPF solution process. Fast start generators are another type of control available to the Optimal Power Flow solution routine. The OPF routine can determine if a generator labeled as a fast start generator would be beneficial in reducing the overall system costs of generation dispatch. If a fast start generator is off-line, but could reduce the cost of the system, then the OPF routine will turn on the generator, and increase the generator's dispatch towards optimizing the system generating cost. Conversely, if a fast start generator is on-line, and the OPF routine determines that reducing the generator's output to 0 would reduce the total generation cost, then the OPF routine will shut off the generator.

Generally speaking, the fast start options should only be used with units with zero Minimum MW limits. Hence it is really aimed at hydro units, or small units which do not need a non-zero minimum MW output for valid operation. This requirement is needed because changing a unit's status is only valid in the OPF routine if it is determined that the unit should dispatch 0 MW to optimize the generating costs of the system.

OPF MW Control

Designates whether the generator's real power output should be included as a control variable in the OPF. This field, which can be toggled, has three possible values:

- "If AGCable" - Generator's control availability depends upon its AGC status.
- "Yes" - Generator is available as a control, regardless of its AGC status.

- "No" - Generator is NOT available as a control, regardless of its AGC status.

Note: in order to be a control the generator must also be in an area or super area on "OPF" control.

Gen MW

The real power output of the generator.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

Cost \$/Hr

The total cost of the generator, including the impact of the cost shift and cost multiplier.

MW Marg. Cost

Tells the marginal cost, in \$ / MWhr, to supply one additional MW of load at this bus. If a generator is available as a control and is not at either its minimum or maximum limit or a cost model breakpoint, then the **MW Marg. Cost** field will be identical to the generator's current marginal cost. However the usual case is for the generator to be at either a limit or a cost model breakpoint so the usual situation is that the **MW Marg. Cost** field values IS NOT equal to the generator's marginal cost.

IC for OPF

Incremental cost of the generator at its current operating point.

Initial MW

The initial real power output of the generator at the beginning of the OPF solution. You can reset the case back to these values by selecting the LP OPF, Restore Previous Control Settings menu item. This menu item is only available following a successful OPF solution.

Initial Cost

The initial generator cost at the beginning of the OPF solution.

Delta MW

Change in the generator's real power output as a result of the OPF.

Delta Cost

Change in the generator's cost as a result of the OPF.

Min MW, Max MW

Minimum and maximum real power output of the generator.

Cost Model

The current cost model being used for the generator. The field value is either "Cubic", indicating that the generator's operating costs are being modeled using a cubic cost function, or "Piecewise Linear", indicating the operating costs are being modeled using a piecewise linear cost function. Toggle the field to change the model. Note that a generator may simultaneously have a cubic model and a piecewise linear model.

Because the OPF uses a linear programming approach, the generator's operating costs are ALWAYS modeled using the piecewise linear model. Generators with an existing cubic cost model are either 1) ignored as OPF controls, or 2) have a piecewise linear cost model automatically created from the cubic model, depending upon the values specified on the [OPF Options and Results Dialog](#).

Cost Curve Points

Shows the number of segments in the piecewise linear model. If no piecewise linear model exists then this field is zero; the generator's costs are being modeled using the cubic function. For such generators you can automatically setup a piecewise linear model simply by entering a non-zero value for the number of points. A piecewise linear model is created that matches as closely as possible the existing cubic model.

Fuel Type

Specifies the fuel type of the generator, if it is known; double-click to toggle through the options. Options are Unknown, Coal, Gas, Hydro, Hydro Pumped, Nuclear, Petroleum, Solar, Wind, and Other.

Profit \$/hr

Shows the profit of the generator. Profit is calculated using this equation:

$$\text{Profit} = (\text{GenMW} * \text{MW Marg Cost}) - [\text{Evaluation of the Generator Cost Function}]$$

OPF Interface Records

Displays OPF specific information about the interface records in the case. To show select **Optimal Power Flow > Results > Interfaces** from the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the Interface records click on the Interfaces tab. This displays the interfaces page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the page.

By default the Interface Records page contains the following fields:

Name

Name of the interface.

Monitor

Specifies whether or not the interface MW limit is enforced in the OPF solution.

Interface MW

The amount of MW flow on the interface.

MW Limit

The interface MW limit.

Percent

The amount of MW flow on the interface as a percentage of the limit.

Monitor Direction

The direction on the interface in which the flow is being monitored.

Monitor Both Directions

If set to **YES**, then the interface flow will be monitored in both directions. Otherwise, it will only be monitored in the **Monitor Direction**.

MW Marg. Cost \$ / MWh

The marginal cost of enforcing the limit on the interface.

Constraint Status

This field will be set to Binding if the interface limit is a binding constraint in the OPF solution, or Unenforceable if the interface limit is unenforceable with the available control set.

OPF Nomogram Records

This list displays OPF specific information about the nomogram records in the case. To show the list, select **Optimal Power Flow > Results > Nomograms** in the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the nomogram records click on the Nomogram Interfaces tab. This displays the nomogram interfaces page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the page.

By default the Interface Records page contains the following fields:

Nomo. Name

Name of the nomogram.

Nomo. Seg.

The segment of the nomogram. Each segment of the nomogram is treated as a separate constraint in the OPF, and therefore is listed individually in the OPF nomogram table.

Monitor

This field will be set to YES if the nomogram segment is monitored during the OPF.

Interface MW Flow

The total MW flow for the segment of the nomogram.

MW Limit

The segment MW limit.

Percent

The amount of MW flow on the segment as a percentage of the segment's limit.

Monitor Direction

The direction on the segment in which the flow is being monitored.

Monitor Both Directions

Yes or No field indicating if both directions should be monitored at the same time.

MW Marg. Cost \$ / MWh

The marginal cost of enforcing the limit on the segment.

Constraint

This field will be set to Binding if the nomogram limit is a binding constraint in the OPF solution, or Unenforceable if the nomogram limit is unenforceable with the available control set.

OPF Line/Transformer Records

Displays OPF specific information about the line and transformer records in the case. To show the display select **Optimal Power Flow > Results > Branches** in the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the Line and Transformer records click on the Line/Transformers tab. This displays the Line/Transformer page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

From Number, From Name, From Area Name

Number, Name, and Area Name of the From bus.

To Number, To Name, To Area Name

Number, Name, and Area Name of the To bus.

Circuit

Circuit identifier for the branch.

Monitor

Specifies whether or not the branch's MVA limit will be enforced in the OPF solution.

Max MVA

The maximum MVA flow on the branch. Value is determined based on the end of the branch with the higher MVA flow.

% of MVA Limit (Max)

The maximum MVA flow as a percentage of the branch MVA limit. Value is determined based on the end of the branch with the higher MVA flow.

Lim MVA

The MVA limit for the branch.

MVA Marg. Cost (\$/MVAhr)

The marginal cost of changing the MVA flow on the branch (i.e., if the line rating were to increase by 1 MVA, what savings could be had).

Constraint

This field will be set to Binding if the branch limit is a binding constraint in the OPF solution, or Unenforceable if the branch limit is unenforceable with the available control set.

OPF DC Lines Records

Displays OPF specific information about the DC lines in the case. To show select **Optimal Power Flow > Results > DC Lines** in the [Model Explorer](#). This display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the same fields as the normal [DC Line Records](#) case information display, with the following additional fields:

OPF Control

Set to **YES** if the DC line is to be controlled during the OPF. To be controlled, at least one of the terminals of the DC line must lie in an area that is on OPF control and this must be set to **YES**.

Min MW or amps, Max MW or amps

The minimum and maximum power (or current) setpoints allowed for the DC line.

OPF Load Records

Displays OPF specific information about each load record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Loads** in the [Model Explorer](#). The OPF Load Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Number, Name

Number and name of the bus to which the load is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for this bus. You can also use the local menu to view the [generator's dialog](#).

ID

Alphanumeric ID used to distinguish multiple loads at the same bus.

Area Name of Load

Name of the load's area.

AGC

Designates whether the load's real power demand is governed by automatic generation control (AGC). If the AGC field is set to Yes the load is on AGC in the standard power flow. When a load is on AGC its real power output is varied automatically during an OPF or SCOPF solution ONLY, provided the load is part of an area or super area that is also on OPF control. Loads will not be dispatched if the area is on some other form of AGC control.

In Simulator OPF the default operating mode is that only loads on AGC control are eligible to be OPF controls. In addition, the load's area or super area must have **AGC Status** of "OPF", and the area must have its Load MW Dispatch set to YES.

MW

The real power output of the load.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Hourly Benefit

The total benefit of the load, including the impact of the cost shift and cost multiplier.

MW Marg. Cost

Tells the marginal cost, in \$ / MWhr, to reduce one additional MW of load at this bus. If a load is available as a control and is not at either its minimum or maximum limit or a cost model breakpoint, then the **MW Marg. Cost** field will be identical to the load's current marginal cost. However the usual case is for the load to be at either a limit or a cost model breakpoint so the usual situation is that the **MW Marg. Cost** field value IS NOT equal to the load's marginal cost.

Inc. Benefit

Incremental benefit of the load at it's current operating point.

Initial MW

The initial real power demand of the load at the beginning of the OPF solution.

Initial Cost

The initial load cost at the beginning of the OPF solution.

Delta MW

Change in the load's real power output as a result of the OPF.

Delta Cost

Change in the load's cost as a result of the OPF.

Min MW, Max MW

Minimum and maximum real power demand of the load.

of Benefit Curve Points

Shows the number of segments in the piecewise linear model. If no piecewise linear model exists then this field is zero.

Profit \$/hr

Shows the profit of the load.

OPF Super Area Records

Displays OPF specific information about each super area record in the case. To show this display select **Optimal Power Flow > Results > Super Areas** in the [Model Explorer](#). The OPF Super Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Super Area

Alpha-numeric identifier of the super area.

AGC Status

Super area's automatic generation control status. This is the same field shown on the Super Area Records display. The field indicates whether or not the super area's generation is changing automatically to control the super area's interchange.

The super area AGC Status field always overrides the AGC Status for the individual areas, except when it is set to "Off AGC".

To be included in the OPF this field MUST be "OPF". In that case the generation costs for all the areas in the super area are included in the OPF objective function. Otherwise they are only included if the super area AGC Status is "Off AGC" and their particular area's AGC Status is on "OPF". Double-click on the field to toggle its value.

Num Areas

Number of areas in the super area. To see the individual areas use the local menu to view the Super Area dialog.

Include Marg. Losses

Specifies whether or not to include marginal losses in OPF calculations for this super area.

MW Marg. Cost Ave.

If the super area is on "OPF" control then for a solved case this field shows the average of the bus MW marginal costs for all the buses in the super area. If there is no congestion then all of the marginal costs should be equal.

MW Marg. Cost St. Dev.

Standard deviation of the bus MW marginal costs for the buses in the super area.

ACE MW

Area control error for the super area.

Gen MW, Load MW

Total real power generation and load in the super area.

Total Sched MW, Int MW

Scheduled and actual interchange real power interchange between the super area and the rest of the system. Both of these fields are the algebraic summation of the scheduled and actual interchange for the areas in the super area.

Loss MW

Total real power losses for the super area.

MW Marg. Cost Min, MW Marg. Cost Max

Minimum and maximum of all the bus MW marginal costs for the buses in the super area.

OPF Controls

The following classes of controls are available during the OPF solution. Note, individual classes of controls can be enabled/disabled for the entire case using the OPF Options and Results dialog and for particular areas using the OPF Area Records display. Also, all classes of controls have associated minimum/maximum limits which are always enforced.

Generator MW output and Fast Start commitment

The generator MW outputs are the major control for controlling the MW flow in the network and for minimizing the objective function. Only generators in areas or super areas that are on "OPF" control are eligible for control; otherwise the generator's MW output remains fixed at its initial value. Whether a particular generator is available for control also depends upon the status of its AGC and OPF MW Control fields. These fields are set on the [OPF Generator Records](#) display. For Fast Start Generators, it is also possible to use the startup and shutdown of these generators as a control. See [OPF Options: Control Options](#) for more details.

Phase shifting Transformer tap position

Phase shifting transformers are used primarily to control the flow of real power in the network. When phase shifting transformers are controlled in the OPF routine, the phase angle is allowed to move anywhere within the phase angle range of the device in order to help alleviate violations on other branches in the system. The flow on the phase shifter is allowed to violate the prescribed MW range given for the phase shifter, but is NOT allowed to violate the MVA rating of the device.

Additionally, in order for a phase shifter to be included in the OPF solution dispatch, the XF Phase property of the area must be set to YES. Each individual phase shifter also has a property for being included in the OPF control that must be turned on in order for the phase shifter to participate in the OPF dispatch. This option can be set for a phase shifting transformer by opening its [Information Dialog](#) and checking the OPF Phase Shifter Control options on the OPF page of the dialog.

Load MW Dispatch

The load MW demands can also be included as controls for re-dispatch during an OPF solution. The concept of controlling a load is generally the same as controlling a generator. Loads can be assigned piecewise linear benefit curves, and included in the OPF dispatch algorithm. Only areas whose AGC control is set to "OPF" are eligible for control. Furthermore, the OPF area's Load MW Dispatch property must also be set to YES. Each area load that is to be included for OPF dispatch must have a benefit model defined, and have its Available for AGC field set to YES. These fields can be set in the [OPF Load Records](#) display.

DC Line MW setpoint

DC Line MW setpoints can be used as controls in the OPF. See [OPF DC Lines Records](#).

OPF Phase Shifter Records

Displays OPF specific information about each phase shifter record in the case. To show this display, select **Optimal Power Flow > Results > Phase Shifters** in the [Model Explorer](#). The OPF Phase Shifter Records display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display. The columns can also be sorted by right-clicking on the heading of the field.

By default the display contains the following fields:

From Number, From Name

The name and number of the bus at the From end of the phase shifter.

To Number, To Name

The name and number of the bus at the To end of the phase shifter.

Circuit

The circuit identifier for the phase shifter.

OPF Control

Specifies whether or not the phase shifter is available for control during an OPF solution.

Area PS Control

Specifies if automatic phase shifter control has been enabled for the area containing the particular phase shifting transformer. If the area phase shifter control is disabled, all phase shifting transformers within the area will remain fixed at their initial settings during the entirety of the OPF solution process. This setting overrides the individual automatic control settings of each phase shifting transformer within the area.

XF Auto

Specifies if the transformer automatic control is enabled. If an individual transformer's automatic control is disabled, it will remain at its initial settings during the entirety of the OPF solution process.

Phase (Deg)

The actual phase shift of the phase shifter, in degrees.

Initial Degrees

The initial phase angle before the OPF solution was calculated.

Delta Degrees

The change in the phase angle during the OPF solution.

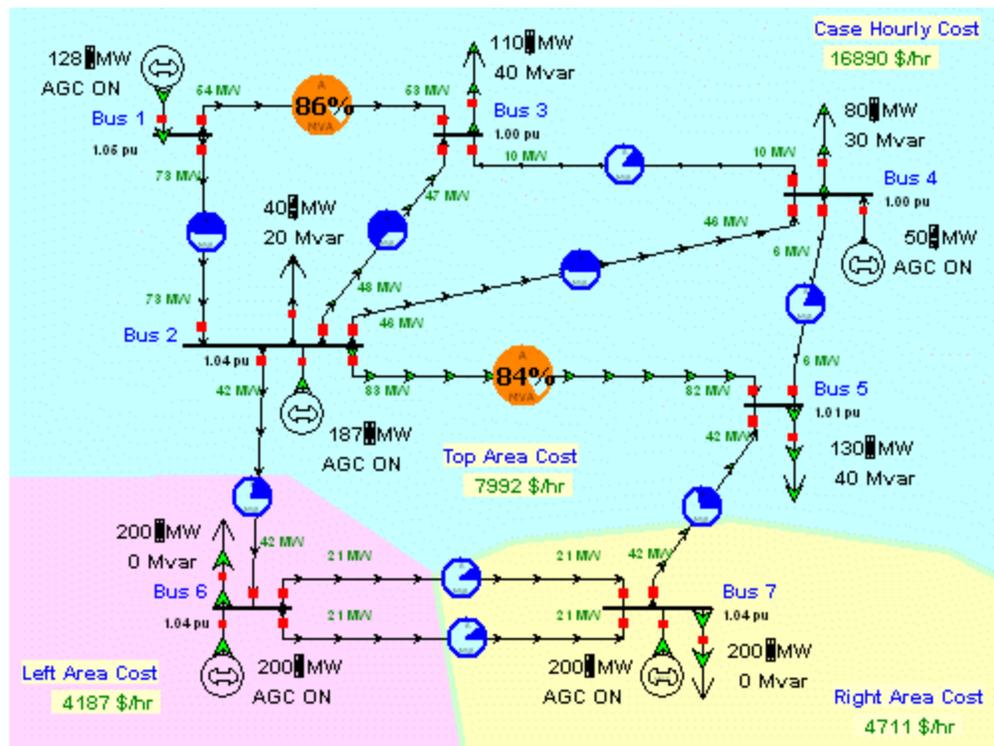
Tap Min, Tap Max

The minimum and maximum tap positions allowed for the phase shifter operation.

OPF Example - Introduction

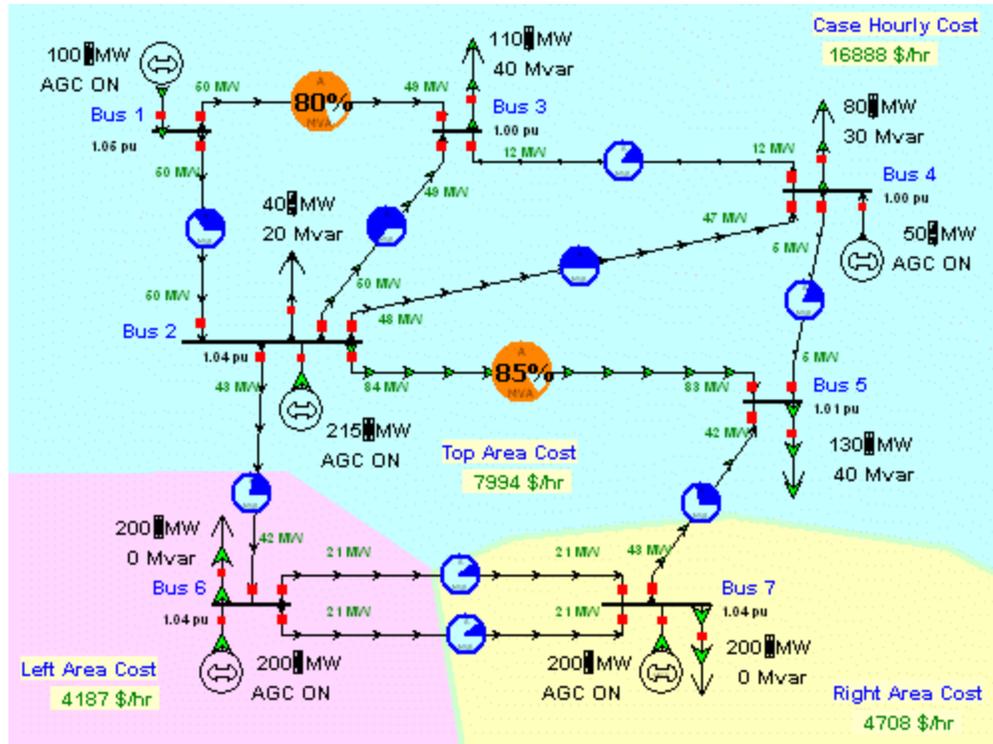
As a simple example of using the OPF, consider the seven bus, three area system contained in the file B7OPF.pwb (included with the PowerWorld Simulator). For this case all three areas are initially on Economic Dispatch (ED) AGC control and hence by default would not be included in the OPF solution. Also, the initial interchange between the areas is equal to zero and the generators are modeled using cubic cost functions.

To initially solve the case using the standard power flow, select **Single Solution** in the **Power Flow Tools** ribbon group on the **Tools** ribbon tab. The case should look like the following figure:



B7OPF Case Solved using Economic Dispatch

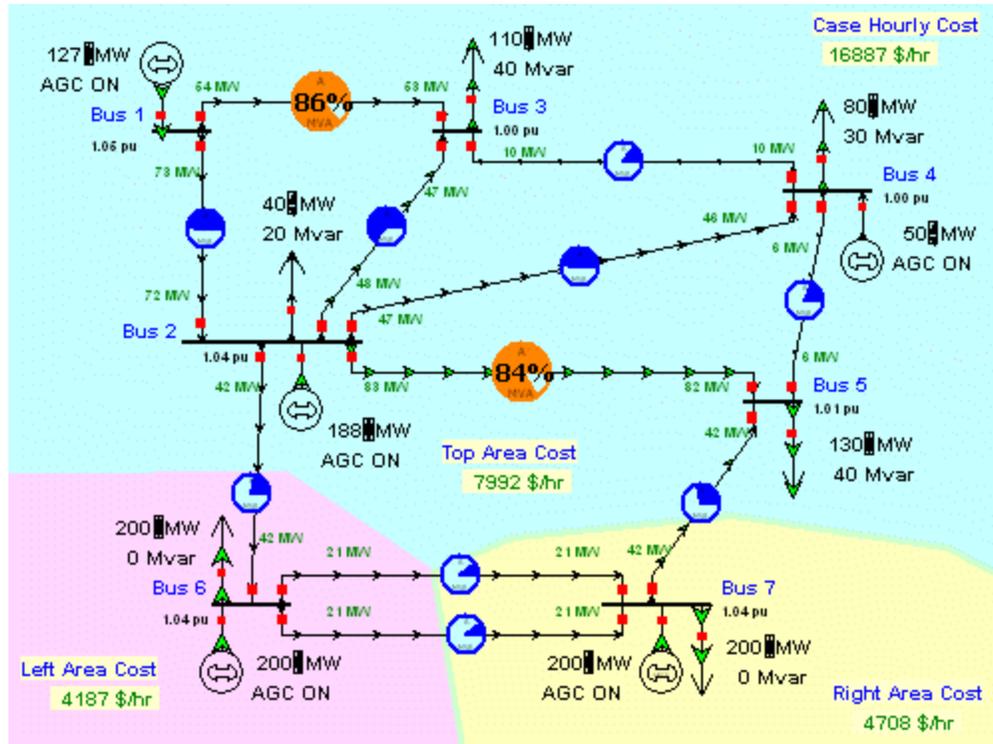
Now we'll modify the case to set the three areas for OPF control. To do this, select **Optimal Power Flow > Results > Areas** in the **Model Explorer** to display the **OPF Area Records display**. Toggle the AGC status for each of the three areas to change it to "OPF". Now select **Primal LP** in the **OPF** ribbon group on the **Add Ons** ribbon tab to solve the case using the LP OPF. The results should look similar to the figure below. The one-line shows the hourly cost for each area and the total case hourly cost, equal to \$ 16,888 / hr.



B7OPF Case Solved using LP OPF

Note that the results are very similar but not identical to the economic dispatch case. We would expect the cases to be similar since for cases with no congestion the OPF solution should be (ideally) equal to the economic dispatch solution. The difference between the two is because in the LP OPF the generator cost functions are converted from a cubic model to a piece-wise linear model using a user specified number of segments, which is 5 segments by default. This value can be viewed/modified from the Control Options page of the [OPF Options and Results display](#).

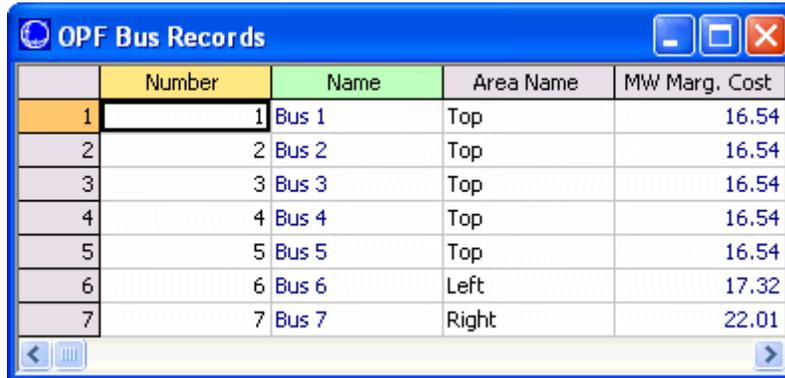
Change the **Total Points Per Cost Curve** field to 100 and resolve. The results are shown in the figure below, which now are almost identical to the economic dispatch results. The disadvantage in using a large number of cost segments is that it degrades the performance of the LP OPF slightly on larger cases.



B7OPF Case Solved using LP OPF with 100 Cost Segments for Generators

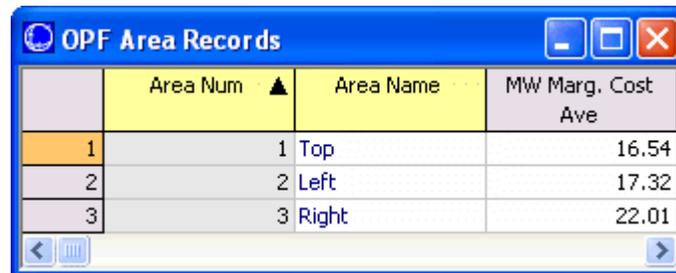
OPF Example - Marginal Costs

Using the OPF solution from the previous page, select **Optimal Power Flow > Results > Buses** in the [Model Explorer](#) to view the bus marginal costs and **Optimal Power Flow > Results > Areas** in the [Model Explorer](#) to view the area marginal costs. The results should be as shown below.



	Number	Name	Area Name	MW Marg. Cost
1	1	Bus 1	Top	16.54
2	2	Bus 2	Top	16.54
3	3	Bus 3	Top	16.54
4	4	Bus 4	Top	16.54
5	5	Bus 5	Top	16.54
6	6	Bus 6	Left	17.32
7	7	Bus 7	Right	22.01

Seven Bus Case Bus Marginal Costs



	Area Num ▲	Area Name	MW Marg. Cost Ave
1	1	Top	16.54
2	2	Left	17.32
3	3	Right	22.01

Seven Bus Case Area Marginal Costs

Note that the marginal costs for all the buses in an area are identical to the area's MW marginal cost. This is the expected result for systems without any line congestion. The area MW marginal costs are not identical. This is because currently each area is independently enforcing its own MW interchange. In the next example we'll jointly dispatch the three areas by combining them into a single [super area](#).

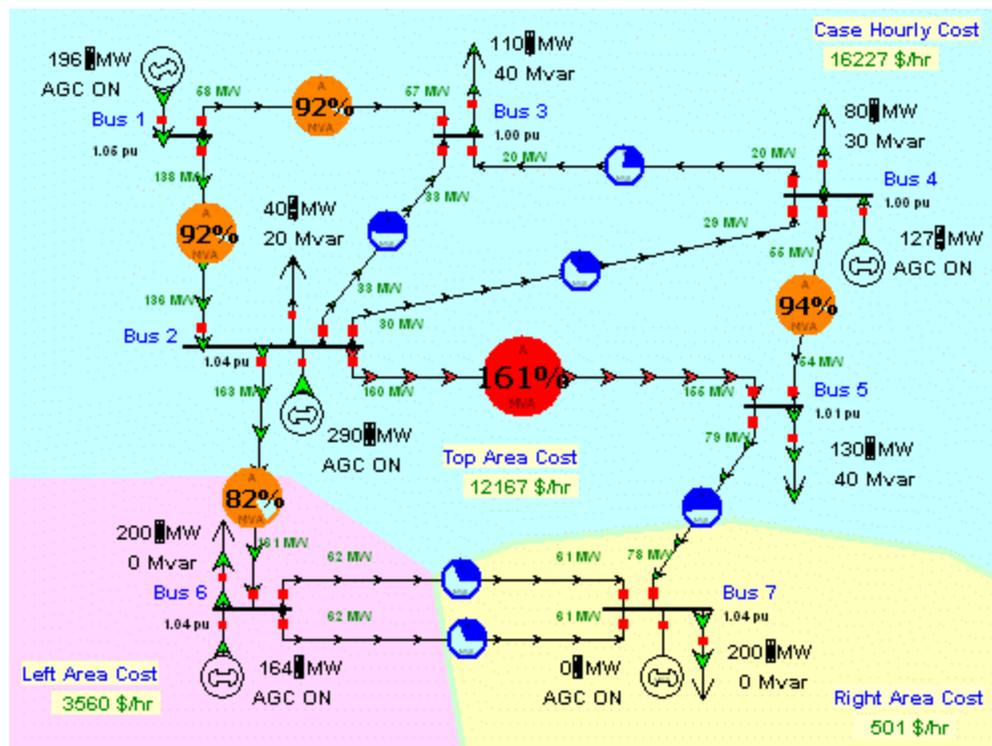
OPF Example - Super Areas

To jointly dispatch the three areas we'll first combine them into a single super area. To setup the super area first select **Optimal Power Flow > Results > Super Areas** from the [Model Explorer](#) to display [the OPF Super Area Records display](#) (alternatively you could also use the **Aggregations > Super Areas** display). To enter a new super area, right click on the "None Defined" entry in the first row of the display to show the display's local menu. Select **Insert**. This displays the [super area dialog](#). Select **Rename** and enter a name for the new super area, "ThreeAreas." To add the three areas to the new super area enter "1-3" in the **New Area #'s** field and then select **Add New Areas by Number** (alternatively you could select the areas from the **New Area Name** list). Also, to enable the super area for control, set the **Super Area Control Options** field (AGC Status) to Optimal Power Flow Control. Select the **Ok** button to save the new super area.

Before resolving the OPF, let's temporarily disable enforcement of line MVA constraints. You can do this from the [Constraint Options page](#) of the [OPF Options and Results dialog](#). Check the **Disable Line/Transformer MVA Limit Enforcement**.

Also, now would be a good time to save the changes. To avoid overwriting the existing B7OPF file, select **Save Case As** from the [Application button](#) to save the case (pwb and pwd files) with a different name, say B7OPFSA (SA for super area).

Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab to resolve the OPF. The results should be as shown below.



OPF Solution with Super Area WITHOUT Enforcing Line MVA Constraints

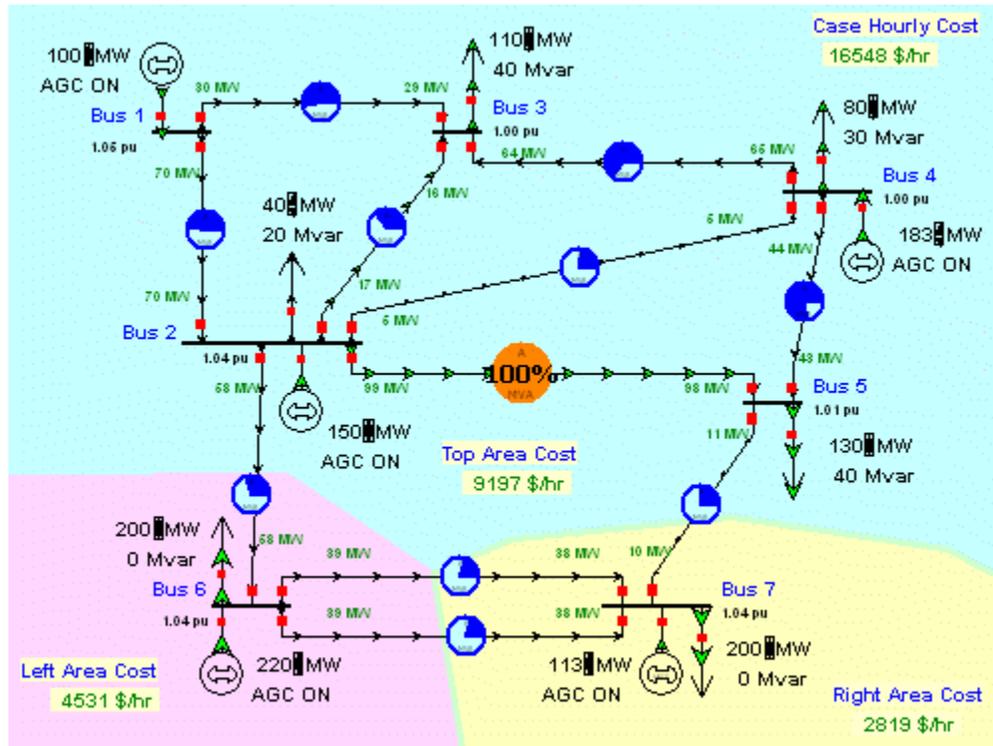
With the super area the individual area interchange constraints are no longer enforced. This permits the free interchange of power between the areas, resulting in an overall decrease in the total case hourly cost from \$ 16,887 / hr to \$ 16,227 / hr. View the OPF Bus Records display to verify that all the bus marginal costs are identical, equal to \$ 17.10 / MWh. Of course the key problem with solving the

system using the super area is that now there are line violations. These violations will be removed next by enforcing the line MVA constraints.

OPF Example - Enforcing Line MVA Constraints

To remove the line MVA violations go back to the [Constraint Options Page](#) of the [OPF Options and Results](#) dialog. Uncheck the **Disable Line/Transformer MVA Limit Enforcement**.

Now resolve the LP OPF, enforcing the line constraints. The resultant solution is shown below.



OPF Solution with Super Area WITH Line MVA Constraint Enforcement

With line constraint enforcement active the OPF optimally redispatches the generation taking into account the line MVA limits. However, enforcing these line constraints comes at a cost. Notice that the total case hourly cost has increased from \$ 16,227 / hr to \$ 16,548 / hr, which is still substantially less than the \$ 16,887 / hr figure we had for the case without the superarea.

Enforcing the line constraints also has an impact on the bus marginal costs, shown below.

Number	Name	Area Name	MW Marg. Cost
1	Bus 1	Top	15.97
2	Bus 2	Top	15.63
3	Bus 3	Top	17.00
4	Bus 4	Top	17.42
5	Bus 5	Top	23.07
6	Bus 6	Left	17.44
7	Bus 7	Right	21.11

Impact of Line MVA Enforcement on the Bus Marginal Costs

The actual marginal cost of enforcing the line constraint can also be viewed on the [OPF Line/Transformer MVA Records](#) display. The MVA Marg. Cost tells the marginal cost of enforcing the

constraint, expressed in units of \$ / MVA / hr. Left-click on the **MVA Marg Cost** field header to sort the display using this field.

OPF Constraints Records

Line/Transformer Constraints

Disable Line/Transformer MVA Limit Enforcement
 Percent Correction Tolerance: 2.0

Only Show Limit Violations
 Only Areas with Line MVA Enforcement
 Use Area/Zone Filters
 Enforce Line/Transformer MW Flow Limits (not MVA)

Interface Constraints

Disable Interface MW Limit Enforcement
 Percent Correction Tolerance: 2.0

Only Show Limit Violations
 Use Area/Zone Filters

Monitor/Enforce Contingent Interface Elements

Never
 Power Flow/OPF but not CA/SCOPF
 All Applications including CA/SCOPF

Do not enforce limits on radials lines and buses
 Limit Monitoring Settings ...
 If you want to change enforcement percentages, modify the Limit Monitoring Settings

Lines/Transformers | Interfaces

	From Number	From Name	From Area Name	To Number	To Name	To Area Name	Circuit	Monitor	Max MVA	% of MVA Limit (Max)	Lim MVA	MVA Marg Cost
1	2	Bus 2	Top	5	Bus 5	Top	1	YES	100.0	100.0	100.0	13.8
2	6	Bus 6	Left	7	Bus 7	Right	1	YES	39.7	19.9	200.0	
3	1	Bus 1	Top	3	Bus 3	Top	1	YES	34.6	53.3	65.0	
4	3	Bus 3	Top	4	Bus 4	Top	1	YES	64.6	64.6	100.0	
5	2	Bus 2	Top	4	Bus 4	Top	1	YES	23.9	23.9	100.0	
6	7	Bus 7	Right	5	Bus 5	Top	1	YES	50.7	25.4	200.0	
7	2	Bus 2	Top	6	Bus 6	Left	1	YES	59.5	29.7	200.0	
8	6	Bus 6	Left	7	Bus 7	Right	2	YES	39.7	19.9	200.0	
9	1	Bus 1	Top	2	Bus 2	Top	1	YES	70.5	47.0	150.0	
10	4	Bus 4	Top	5	Bus 5	Top	1	YES	45.6	76.0	60.0	
11	2	Bus 2	Top	3	Bus 3	Top	1	YES	28.1	35.1	80.0	

Security Constrained Optimal Power Flow Overview

Note: The SCOPF option in PowerWorld Simulator is only available if you have purchased the SCOPF and OPF add-ons to the base package. To learn more about the SCOPF, please feel free to read through the information contained in these help files. Contact PowerWorld Corporation for details about ordering the SCOPF and OPF versions of Simulator.

The [optimal power flow](#) (OPF) algorithm has the purpose of minimizing an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. Normally however, the secure operation of a power system requires that there be no unmanageable contingency violations. Thus, the minimization of the objective function requires considering contingencies. This is achieved using a security constrained optimal power flow (SCOPF) algorithm.

While the [optimal power flow](#) (OPF) algorithm determines the optimal state of the system by iterating between solving a power flow and solving a linear program that changes the system controls, during the [SCOPF solution process](#) in addition to solving the base case power flow, all contingencies must also be solved. The linear program which then runs changes the system controls to remove contingency violations as well as base case operating violations.

The SCOPF algorithm makes control adjustments to the base case (pre-contingency condition) to prevent violations in the post-contingency conditions. If enough controls are available in the system, the solution minimizes the [objective function](#) and the system enforces contingency violations. If the system does not have enough controls, then some violations may be persistent under certain contingencies. Those represent unenforceable constraints, which result in high bus marginal costs.

The commands and options for the SCOPF are accessed by selecting the **SCOPF** option from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The SCOPF function uses the OPF options defined in the [OPF Options and Results dialog](#) and the contingency settings specified in the [contingency analysis dialog](#). Please read through the OPF and the contingency analysis help for more information about these settings. We encourage you to become familiar with Simulator OPF and the contingency analysis tool before running SCOPF simulations. Other commands in the [SCOPF dialog](#) are used to specify the base case solution process and for accessing the SCOPF results.

SCOPF Objective Function

The SCOPF objective function uses the function defined in the OPF settings. There are two objective functions in Simulator: Minimum Cost and Minimum Control Change. Minimum Cost attempts to minimize the sum of the total generation costs in specified areas or super areas. Minimum Control Change attempts to minimize the sum of the absolute value of the change in the generation in the specified areas or super areas. The objective function is set up in the [OPF Options and Results dialog](#).

The result of the SCOPF will be different from the OPF solution because the SCOPF meets additional [inequality constraints](#) associated with the contingency violations.

SCOPF Dialog

The SCOPF dialog allows the user to control options of Security Constrained OPF, as well as to access the optimization results.

The SCOPF can be run from this dialog using the **Run Full Security Constrained OPF** button at the top of the form.

There are three pages of information included in this form:

[Options](#)

[Results](#)

[LP Solution Details](#)

SCOPF Solution Process

The SCOPF involves three major steps that can be solved either automatically or manually from the SCOPF control dialog:

- Initialization to setup the SCOPF LP tableau and the control structures
- Contingency analysis calculation and storage of control sensitivities associated with each contingency violation
- SCOPF iterations, which include an LP solution and a power flow solution. During each LP step in the LP routine, the algorithm enforces the newest most severe contingency violation. After each violation is processed, all of the unprocessed violations are updated. This step is crucial since often resolving the most severe violation resolves numerous other violations. For instance, a single line might be overloaded in a number of contingencies: fixing the worst contingency fixes the others as well. On the other hand, processing some violations may result in new violations. In order to verify that no new violations have been created by the control changes made, the SCOPF will go back to step two and reprocess all the contingencies and the new base solution. The number of times this iteration occurs is determined by the Maximum Number of Outer Loop Iterations.

The SCOPF terminates when all the contingency violations have been processed. Note that the user can rerun the SCOPF by repeating the solution process if they want to verify the contingency violation enforcement at the new optimal operating point.

SCOPF Results

The Results page of the [Security Constrained Optimal Power Flow](#) form provides information on the results of the latest SCOPF solution, including the [contingency violations](#) included, the [marginal price details](#), and the [marginal control details](#).

SCOPF Equality and Inequality Constraints

Two general types of constraints are involved in the SCOPF solution: equality and inequality constraints. Equality constraints are constraints that have to be enforced. That is, they are always "binding". For example in the SCOPF, as well as in the OPF and in the power flow, the real and reactive power balance equations at system buses must always be satisfied (at least to within a user specified tolerance). In contrast, inequality constraints may or may not be binding. For example, a line MVA flow under a certain contingency may or may not be at its limit.

The SCOPF problem is solved by iterating between a power flow solution and a contingency constrained LP solution, some of the constraints are enforced during the power flow solution and some constraints are enforced during the LP solution. The constraints enforced during the power flow are, for the most part, the constraints that are enforced during any power flow solution. These include the bus power balance equations, the generator voltage set point constraints, and the reactive power limits on the generators. What differentiates the SCOPF from a standard power flow and from the OPF are the constraints that are explicitly enforced by the LP solver. These include the following constraints:

[Equality Constraints](#)

[Inequality Constraints](#)

SCOPF Equality Constraints

The SCOPF equality constraints are the same as the [OPF equality constraints](#): Area MW interchange, bus MW and Mvar power balance, Generator voltage setpoint and super area MW interchange.

SCOPF Inequality Constraints

The following classes of inequality constraints are enforced during the SCOPF solution.

Generator real power limits

Generator real power limits are enforced during the SCOPF LP solution.

Generator reactive power limits

Generator reactive power limits are enforced during the SCOPF LP solution.

Interface MW Limits

Interface MW limits are enforced during the SCOPF solution. [Interfaces](#) are used to represent the aggregate flow through a number of different devices. During the SCOPF the MW post-contingency flow through the interface is constrained to be less than or equal to a user specified percentage of its limit, provided the interface is active for enforcement. For an interface to be active for enforcement the following three conditions must be met:

- Interface enforcement must not be disabled for the case. This field can be set from either the [OPF Constraints Dialog](#) or the [OPF interfaces records](#). As default, the interface enforcement is not disabled. Note that interface flow is limited to a percent of its limit as specified by the interface's [Limit Monitoring Settings](#).
- Interface enforcement must be active for at least one of the interface's areas. Note, an interface is assumed to be in each area that contains at least one of its components. This field can be set from the [OPF Area Records display](#). Note: the default is that interface enforcement is not active, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual interface. This field can be specified from the [OPF interfaces records](#) display or in the [Limit Monitoring Settings](#) dialog. The default is active.

Each interface that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of contingency enforcement. When manually solving the SCOPF one can skip a contingency violation associated to the interface by setting the Include field of the [SCOPF CTG Violations](#) dialog to **No**.

Transmission Line and Transformer (Branch) MVA Limits

Transmission line and transformer (branch) MVA limits are enforced during the SCOPF solution. During the LP the post-contingency branch line flow is constrained to be less than or equal to a user specified percentage of its limit, provided the branch is active for enforcement. For a branch to be active for enforcement the following three conditions must be met:

- Line/Transformer enforcement must not be disabled for the case. This field can be set from either the [OPF constraints dialog](#) or the [OPF Line/Transformer Records](#) display. The default is that case line/transformer enforcement is not disabled. Also note that the branch flow is limited to a percent of its limit as specified by the branch's [Limit Monitoring Settings](#).
- Branch enforcement must be active for the branch's area. Enforcement for tie-lines must be active for either area. This field can be set from the [OPF Line/Transformer Records](#) display. The default is that branch enforcement is not active, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual branch. This field can be set from the [OPF Line/Transformer Records](#) display or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each branch that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding under contingency conditions. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcing the contingency constraint, which is shown on the [SCOPF Bus Marginal Price Details](#) dialog. When manually solving the SCOPF one can skip a contingency violation associated to the branch by setting the Include field of the [SCOPF CTG Violations](#) dialog to **No**.

SCOPF Control

The control dialog allows the user to manually or automatically run a SCOPF simulation and visualize the execution of the simulation. The [SCOPF solution process](#) involves three steps: base case solution and initialization, contingency analysis, and SCOPF iterations.

Run Full Security Constrained OPF

Press this button to run the three steps of the SCOPF solution automatically. The SCOPF will solve the base case using the selected method, will use the currently stored list of contingencies during the contingency analysis step, and will take the CA results and sensitivities to iterate in order to obtain the optimal solution that minimizes cost and enforces contingency violations.

Options

Maximum Number of Outer Loop Iterations

Indicates the number of maximum outer loop iterations. The outer loop iterations determines how many times the contingency analysis will be re-run following a successful SCOPF dispatch. In this manner, Simulator will look for new violations that may occur due to the new generation dispatch.

Consider Binding Contingent Violations from Last SCOPF Solution

When checked, this option ensures that the contingent violations from the last SCOPF solution are included in the current SCOPF solution. This option is helpful in preventing the SCOPF from hunting between having a constraint binding in one solution, and resolving with it not binding in a later solution because it was previously remedied. This option should generally always be checked, unless the user is sure that the previous solution has no bearing on the current solution, such as having made major changes to the system since the previous solution.

Initialize SCOPF with Previously Binding Constraints

When checked, this option results in the SCOPF solution process starts with the exact same LP tableau from the last solution. This can make for fairly fast solutions (recognizing that the contingency analysis needs to be resolved) when the changes to the system are small. Simulator automatically uses this option when doing multiple outer loops of the SCOPF. This option allows the user to solve the outer loops (set the outer loop counter to 1) by repeatedly solving the SCOPF manually, potentially making modifications between solutions if desired.

Set Solution as Contingency Analysis Reference Case

Check this field to set the solution of the SCOPF as the contingency analysis reference. If the system has enough controls to remove all the contingency violations, a rerun of the contingency [analysis](#) using the SCOPF solution as the reference should report no branch violations.

Maximum Number of Contingency Violations Allow Per Element

Specify the maximum number of contingency violations that the SCOPF analysis should allow per element. After completing the power flow solutions for the base case and the contingencies, only this number of contingency violations per element are then passed to the linear programming algorithm. The assumption is that if you fix the 10 worst contingency violations for an element then the others will also be fixed.

Basecase Solution Method

Specifies whether the solution of the base case is performed using the power flow algorithm or the optimal power flow algorithm. The selection will affect the initial conditions of the system and consequently the contingency analysis results and the sensitivities used by the LP solver. Currently

the SCOPF does not resolve the contingency analysis during the optimization since this is computationally expensive. See the [SCOPF solution process](#) for details.

Handling of Contingent Violations Due to Radial Load

It is often common when computing a security constrained OPF to have violations occur on branches due to radial load. In these instances, there is no way to adjust controls to continue to serve the load, without overloading the serving element. Therefore you can choose how contingent violations of this type should be handled by the SCOPF. You can choose to flag them but not include them in the SCOPF, ignore them completely, or include the violations in the SCOPF. Note that if you include the violations in the SCOPF, the SCOPF algorithm will not be able to remove the violation on the element via generation dispatch. However, it may be able to do so if the load in question has a load benefit curve defined and is available for load shed dispatch in the SCOPF routine.

DC SCOPF Options

The DC options given for the SCOPF revolve around the treatment of Line Outage Distribution Factors (LODF) during the DC SCOPF solution. You can choose to discard the LODFs when the SCOPF is finished, store them in memory (lost when Simulator is closed,) or in memory and in the PWB file (if saved.)

If the LODFs have been stored and you wish to clear them (they can require quite a bit of RAM depending on the number of contingencies and size of the case,) you can press the button labeled **Clear Stored Contingency Analysis LODFs**.

SCOPF Results Summary

Number of Outer Loop Iterations

The number of outer loop iterations required to solve the SCOPF.

Number of Contingent Violations

Number of violations from the contingency analysis portion of the solution, which are used to attempt to determine the security constrained dispatch.

SCOPF Start Time, SCOPF End Time

Physical time when the SCOPF solution process started and finished.

Total Solution Time (Seconds)

Length of time needed to determine the SCOPF solution.

Total LP Iterations

Total number of Linear Programming iterations necessary to determine the SCOPF solution.

Contingency Analysis Input

Number of Active Contingencies

The number of contingencies included in the SCOPF simulation. Specific contingencies may be excluded from the simulation in the [contingency analysis dialog](#) by changing the skip field of a contingency to YES. The contingency analysis dialog can be conveniently accessed from the SCOPF control dialog.

View Contingency Analysis Form

Clicking this button shows the [contingency analysis dialog](#).

Contingency Analysis Results

This window allows the user to monitor the details of the contingency analysis step. During contingencies, the outage actions, the solution of each contingency, and the solution of the CA run are reported.

SCOPF CTG Violations

The contingency violations page is available from the Results tab of the [Security Constrained Optimal Power Flow](#) form. The display lists the results from the latest contingency analysis run including the violations that were included in the SCOPF and the final error for each violation. This dialog may change if the user reruns the SCOPF by solving the contingency analysis using the SCOPF solution as contingency analysis reference. Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display shows the following fields:

Contingency Name

This is the contingency label. By default single line contingencies start with an "L", single generator outages with a "G", and single transformer outages start with an "X".

Category

Currently, the SCOPF considers only branch and interface violations. Thus, the category of the contingencies should be branch MVA or Interface.

Element

Shows information about the specific element that presented the violation. When the violation occurs in a branch, this column includes the identifiers of the sending and receiving ends of the branch, the circuit, and the direction of the violating flow. Since Interfaces are directed, this field will present only the interface name in the case of violating interfaces.

Value

The percentage flow that appears in the branch during the contingency prior to optimization. If this number is larger than the scaled limit, the violation needs to be removed.

Scaled Limit

The scaled limit corresponds to the Line/Transformer Percentage specified in the [limit monitoring settings](#) dialog. By specifying this limit to be higher than 100% some of the contingency violations might be effectively relaxed. Sometimes this helps the OPF and the SCOPF obtain a feasible solution. On the other hand, it is often required to analyze the performance of the system if branches would have higher ratings.

New Value

The percentage flow that appears in the branch during the contingency after SCOPF optimization. If this number is larger than the scaled limit, the contingency violation has not been removed and it is therefore unenforceable. If the value is equal to the scaled limit, then the contingency violation constraint would be binding. If the value is smaller than the scaled limit, the contingency violation has been removed.

Error

The difference between the new value and the scaled limit. If the error is positive, the line is unenforceable. If the error is zero, the constraint has been corrected.

Included

Indicates if the contingency violation was included as a constraint in the SCOPF solution.

Marginal Cost

Indicates the cost associated with the contingency violation. If the constraint is unenforceable, the marginal cost is assigned arbitrarily as a high value in the [OPF constraint options](#) dialog.

Unenforceable

Indicates whether the contingency violations is unenforceable, i.e., the system has not enough controls to relieve the branch overload when the contingency occurs.

Skip Violation?

Change this field to NO if the contingency violations should not be included as a SCOPF constraint. This is sometimes useful in order to analyze the effect of the contingency violation in the SCOPF solution. This field may be toggled when doing a manual SCOPF solution.

SCOPF LP Solution Details

The LP Solution Details page of the [Security Constrained Optimal Power Flow](#) form provides information on the linear program solution of the SCOPF, including a list of [All LP Variables](#), [LP Basic Variables](#), and [LP Basis Matrix](#). This information applies to the linear programming tableau solution method.

SCOPF All LP Variables

The SCOPF All LP Variables dialog displays the basic and non-basic variables associated with the final LP SCOPF solution. Users interested in the specifics of the LP SCOPF can access this page to obtain internal information about the SCOPF solution. To see the display, open the [Security Constrained Optimal Power Flow](#) form, click on the LP Solution Details tab, and access the All LP Variables page.

Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the LP variables with the following fields:

ID

Variable identifier.

Org. Value

The initial value of the LP variable before SCOPF optimization.

Value

The final value of the LP variable after SCOPF optimization.

Delta Value

The difference between the original value field and the value field.

Basic Var

Shows the index of the basic variables in the LP basis. If the value is zero, the variable is non-basic. These values are set up after the SCOPF calculates the contingency violation sensitivities.

NonBasicVar

Shows the index of the non-basic variable. If the value is zero, the variable is basic.

Cost(Down)

The cost associated with decreasing the LP variable. The field will show if the variable is at its max or min limit.

Cost(Up)

The cost associated with increasing the LP variable. The field will show if the variable is at its max or min limit.

Down Range

The available range to decrease the basic variable before a new constraint is hit under a contingency condition.

Up Range

The available range to increase the basic variable before a new constraint is hit under a contingency condition.

Reduced Cost Up

The cost reduction that would be experimented if a LP variable increases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

Reduced Cost Down

The cost reduction that would be experimented if a LP variable decreases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

At Breakpoint

Yes, if the LP variable is at a break point.

SCOPF LP Basic Variables

The LP Basic Variables (from the SCOPF [LP Solution Details](#) page) displays the basic variables of the final LP solution. The basic variables may correspond to controls that can be altered to minimize the objective function, or slack variables associated with unenforceable constraints. Users interested in the specifics of the LP SCOPF can access this page to obtain internal information about the SCOPF solution. Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each LP variable with the following fields:

ID

Basic variable identifier.

Org. Value

The initial value of the basic LP variable before the SCOPF optimization.

Value

The final value of the basic LP variable after the SCOPF optimization.

Delta Value

The difference between the original value field and the value of the basic variable.

Basic Var

Shows the indices of the basic variables in the LP basis.

Cost(Up)

The cost associated with increasing the basic variable.

Down Range

The available range to decrease the basic variable before a new constraint is hit under a contingency condition.

Up Range

The available range to increase the basic variable before a new constraint is hit under a contingency condition.

SCOPF LP Basis Matrix

The LP Basis Matrix page of the SCOPF [LP Solution Details](#) tab displays the basis matrix associated with the final SCOPF LP solution. There is one row per constraint and one column per basic variable. Additional columns summarize information associated with each constraint. This page is usually only of interest to users interested in knowing the specifics of the SCOPF solution. Knowing the basis matrix can be helpful in figuring out why a particular SCOPF solution exhibits a certain behavior. The entries in the basis matrix give the sensitivity of each constraint to each of the basic variables.

As any case info display in simulator, right click to see options to copy information to the clipboard and to perform standard windows actions, such as printing.

SCOPF Bus Marginal Price Details

This display (from the SCOPF [Results](#) page) shows information about the components of the marginal cost at each bus. The display is relevant to see how the bus marginal cost depends on the cost of enforcing system constraints such as branch limits and area equality constraints. This display is useful for indicating which constraints are contributing towards the determination of the marginal price at each bus.

As any case info display in Simulator this display can be customized and the information copied, printed, and saved by accessing the local menu option with the mouse right click.

One key feature of this display is the breakdown in the bus MW marginal cost into three components: the cost of energy, the cost of congestion, and the cost of losses. Also, while the bus MW marginal cost does not change based on the reference, the costs of energy, congestion, and losses are reference dependent. This reference is set on a per-area (or per-super area) basis in the [Area information dialog OPF tab](#) (or the [Super Area information dialog OPF tab](#)).

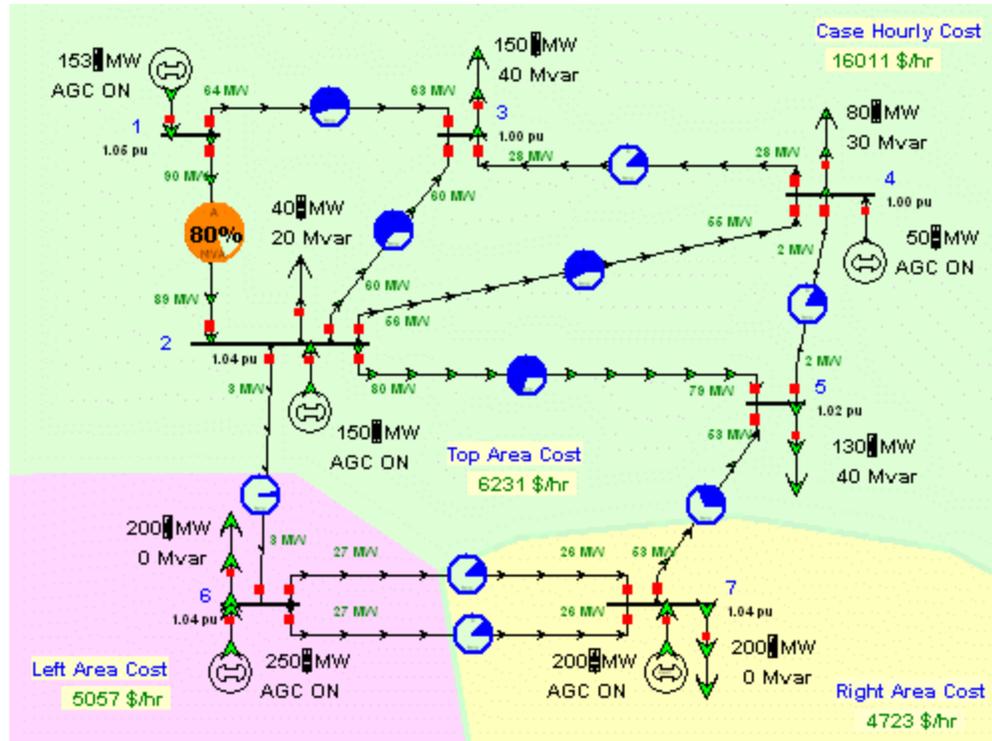
SCOPF Bus Marginal Controls

This display (from the SCOPF [Results](#) page) shows the sensitivities of the controls with respect to the cost at each bus. A change in a system control will have the indicated effect in the marginal cost at the system buses. Vice-versa, the marginal cost at a bus is affected by changes in the value of the basic variables.

The Marginal Controls page can be accessed by opening the [Security Constrained Optimal Power Flow](#) form, and clicking on the Results tab.

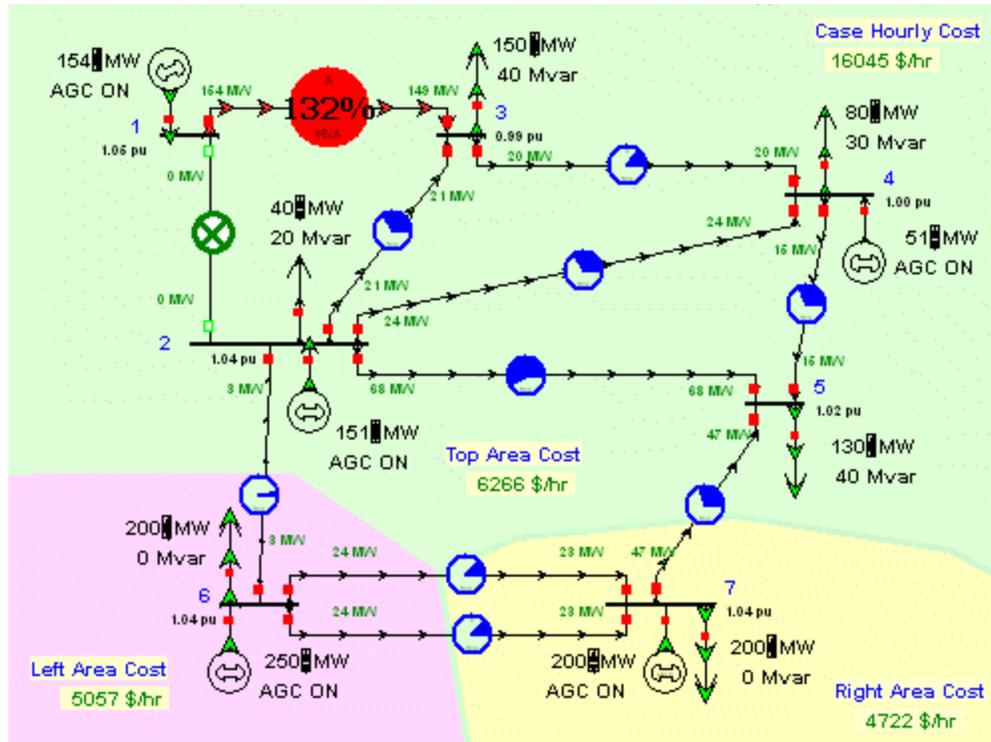
SCOPF Example: Introduction

In this section we introduce an example of using the SCOPF. Consider the seven bus, three area system contained in the file B7SCOPF (included with the PowerWorld Simulator). This case is the same case used in the OPF example, except that the line from bus 2 to 3 has now a 100MVA rating, lines 1 to 2, 1 to 3 and 2 to 5 have a 120MVA rating, and all three areas are initially on OPF control. To initially solve the case using the optimal power flow, select **Primal LP** in the **OPF** ribbon group on the **Add Ons** ribbon tab. The solution obtained is shown below. The total case hourly cost is \$16,011 / hr.



B7SCOPF Case Solved using OPF

We are interested in determining an optimal solution that meets security constraints under contingency conditions. In order to show that the current OPF solution does not enforce contingency violations, take line 1 to 2 out of service by clicking in a circuit breaker. Then solve the power flow by pressing the **Single Solution** button. The result indicates that line 1 to 2 is overloaded 32%. The SCOPF algorithm will attempt to move the operating solution such that no contingency violation occurs in the system. Close line 1 to 2 back in service.

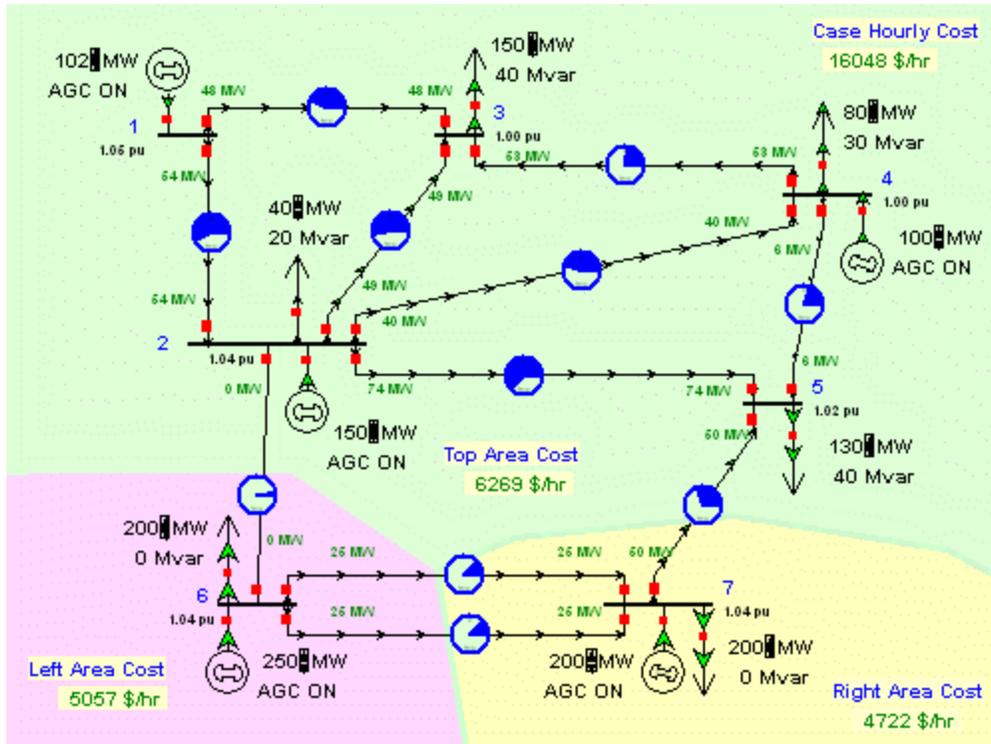


B7SCOPF Power Flow Solution with line 1-2 open

The next step is to specify the contingency conditions that the system should withstand. In order to do that, we access the [Contingency Analysis dialog](#) under **Tools**. Note that we can also access this dialog from the [SCOPF control dialog](#) by selecting **Add Ons > Security Constrained OPF** and pressing the **View Contingency Analysis Form** button on the **Options** page. The B7SCOPF case does not have contingencies associated with it. Insert single line contingencies in the contingency list by pressing the **Auto Insert** button located at the bottom left of the Contingency Analysis dialog. In the [Auto Insert Dialog](#) select the option for **Single Transmission Line or Transformer** and select **Numbers** under the *Identify buses by* field. Select **Do Insert Contingencies**. This will prompt to insert 11 single line contingencies corresponding to all the lines in the system. Select **Yes**. You can now close the Contingency Analysis dialog.

Return to the **Security Constrained Optimal Power Flow Form**. You can set the SCOPF to use the OPF solution as the base by selecting **Solve base case using optimal power flow**.

You can now solve the SCOPF by pressing the **Run Full Security Constrained OPF**. This will process the contingency violations and iteratively solve the LP program and the power flow equations to minimize the objective function and enforce equality and inequality constraints. The solution is shown below. The total operating cost is now \$16,048 /hr. The increase in operating cost is due to enforcing security constraints. If a new contingency analysis is performed using the optimal solution as the reference, it will be found that no contingency violations occur for the contingencies in the list, i.e., branch flows are less than (or equal to) 100% in the post contingency condition. Thus, the system meets all the specified constraints. You can analyze the SCOPF results by browsing the information in the SCOPF tabs. Note that the **CTG Constraints** dialog does not show unenforceable constraints, but the branch violation of line 2 to 5 due to the contingency 5 to 7 is now binding.



B7SCOPF Case Solved using SCOPF: The system now meets the contingency constraints.

SCOPF Example: Marginal Prices

Using the SCOPF solution from the previous page, select the **Bus Marginal Price Details** from the **Results** page of the **SCOPF Dialog** to view the detail of the marginal price components. Note in the following Figure that each area constraint contributes equally to the marginal cost of the buses in that area. The binding inequality constraint from bus 2 to 5 makes further contribution to the bus marginal price of buses in area Top.

	Number	Name	Area Name	MW Marg. Cost	Energy \$/MWh	Congestion \$/MWh	Losses \$/MWh	Area 1 MW Constraint	Area 2 MW Constraint	Area 3 MW Constraint	Line from 1 to 3 ckt. 1
1	1	1	Top	13.00	12.85	0.15	0.00	12.85	0.00	0.00	0.00
2	2	2	Top	12.85	12.85	0.00	0.00	12.85	0.00	0.00	0.00
3	3	3	Top	13.75	12.85	0.90	0.00	12.85	0.00	0.00	0.00
4	4	4	Top	14.00	12.85	1.15	0.00	12.85	0.00	0.00	0.00
5	5	5	Top	17.82	12.85	4.96	0.00	12.85	0.00	0.00	0.00
6	6	6	Left	17.60	17.60	0.00	0.00	0.00	17.60	0.00	0.00
7	7	7	Right	21.80	21.80	0.00	0.00	0.00	0.00	21.80	0.00

Seven Bus Case SCOPF Bus Marginal Price Details

SCOPF Example: Unenforceable Constraints

Using the same example as above, reduce the MVA rating of line 2 to 5 to 90 MVA. The initial base case LP OPF solution is the same as in the previous example. Consider the same contingency list. When the contingencies are solved using the initial OPF solution as the reference, seven contingency violations need to be removed. The CTG dialog after simulation is presented in the following figure.

The screenshot shows the 'Security Constrained Optimal Power Flow Form' dialog box. The 'SCOPF Status' is 'SCOPF Solved Correctly'. The 'LP Solution Details' tab is active, showing a table of 'Contingency Violations'. The table has columns for Contingency Name, Element, Value, Scaled Limit, New Value, Error, Included, Marginal Cost, and Unenforceable. Contingency 7 is highlighted in yellow, indicating it is unenforceable.

	Contingency Nam ▲	C	Element	Value	Scaled Limit	New Value	Error	Included	Marginal Cost	Unenforceable
1	L_00001-00002C1		Br 1 (1) -> 3 (3) CKT 1 at 1	157.31	120.00	-2.76	0.00	NO	0.00	NO
2	L_00001-00003C1		Br 1 (1) -> 2 (2) CKT 1 at 2	154.45	120.00	10.45	0.00	NO	0.00	NO
3	L_00002-00003C1		Br 2 (2) -> 5 (5) CKT 1 at 2	90.73	90.00	68.68	0.00	NO	0.00	NO
4	L_00002-00004C1		Br 2 (2) -> 5 (5) CKT 1 at 2	92.52	90.00	66.18	0.00	NO	0.00	NO
5	L_00006-00007C1		Br 2 (2) -> 5 (5) CKT 1 at 2	91.60	90.00	73.24	0.00	NO	0.00	NO
6	L_00006-00007C2		Br 2 (2) -> 5 (5) CKT 1 at 2	91.60	90.00	73.24	0.00	NO	0.00	NO
7	L_00007-00005C1		Br 2 (2) -> 5 (5) CKT 1 at 2	127.56	90.00	104.48	14.48	YES	1000.00	YES

CTG Dialog after SCOPF Solution

We note that the constraint from bus 2 to 5 under a contingency from 5 to 7 is unenforceable. There are not enough system controls to enforce the contingency constraint. A \$ 1,000 / hr cost is assigned to unenforceable constraints in this case. The cost of not enforcing constraints can be specified in the [OPF Constraint Options](#) Dialog.

Optimal Power Flow Reserves Overview

Note: The OPF Reserves tool in PowerWorld Simulator is only available if you have purchased the OPF Reserves and OPF add-ons to the base package. To learn more about the OPF Reserves, please read through the information contained in these help files. [Contact PowerWorld Corporation](#) for details about ordering the OPF and OPF Reserves version of Simulator.

In order to prevent load disconnections and loss of stability during normal operation or in the case of unexpected events, power systems should operate with an adequate level of Reserve. Reserve is an ancillary service needed for the successful operation of the system and the electricity market. This service can be obtained in a regulated, mandatory manner, or it can be provided by an Ancillary Services Reserves Market. Simulator OPF Reserves is the tool used to simulate Ancillary Services Reserve Markets.

In the Reserves Market, the generators (and sometimes loads) supply bids to sell the ability to take demand (increase their output) in a fast manner if called to do so. While an energy-only market has only one product, active energy, the Reserves Market includes the energy product and several reserve services: regulating, spinning and supplemental reserve. Similar to electricity markets for energy, which deal with active power, the reserve market focuses only on active power reserves.

The optimal power flow (OPF) algorithm by itself is able to simulate energy-only electricity markets by determining the minimum cost or minimum control change dispatch subject to normal operation constraints. The OPF Reserves considers special OPF Reserve Constraints at the area and zone level, and OPF Reserve Controls provided by generators or loads. OPF Reserves will thus simultaneously co-optimize energy and reserve and maximize total social surplus producing both energy (LMP) and Reserve Marginal Clearing Prices (RMCP).

The commands and options for the [OPF Reserves](#) are integrated with the [OPF tool](#).

OPF Reserves Topics

The commands and options for the OPF Reserves are integrated with the [OPF tool](#). The OPF Reserves option is activated from the [Common Options tab](#) of the OPF Options and Results dialog by checking the **Include OPF Reserve Requirements** box.

The following topics provide details about how the OPF Reserves are integrated into the OPF tool and options and input data can be set:

- [OPF Reserves Controls](#)
- [OPF Reserves Constraints](#)
- [OPF Reserves Objective Function](#)
- [OPF Reserves Results and Pricing](#)
- [OPF Reserves Case Information Display](#)
- [Area and Zone Reserve Requirements Curves](#)
- [Generator and Load OPF Reserve Bids](#)
- [OPF Reserves Example](#)

OPF Reserves Controls

Generators can provide the following types of reserve:

- Regulation Reserve (RR) is provided by online fast units, usually tied to AGC primary control, and capable to regulate the small positive or negative imbalances caused by the random nature of loads.
- Spinning Reserve (SR) is provided by online units and is used to take larger variations of load and losses and unexpected events.
- Supplemental Reserve (XR) is provided by online or offline fast-start units. This type of reserve is used to correct large imbalances caused by contingencies.

Spinning and supplemental reserve are positive quantities. Regulation reserve is a bidirectional control. There are two ways to model regulation controls: As two independent controls: regulation reserve up (RR⁺) and regulation reserve down (RR⁻). As a single control: in this case the unit provides the same amount of regulating reserve in both directions. In order to tell Simulator whether to use a single bidirectional control for regulating reserve up or down, go to the [OPF Options and Results Dialog](#) and check the **Use a Single Control for Up/Down Regulation** box.

Spinning and supplemental reserves combined together provide Contingency Reserve (CR)

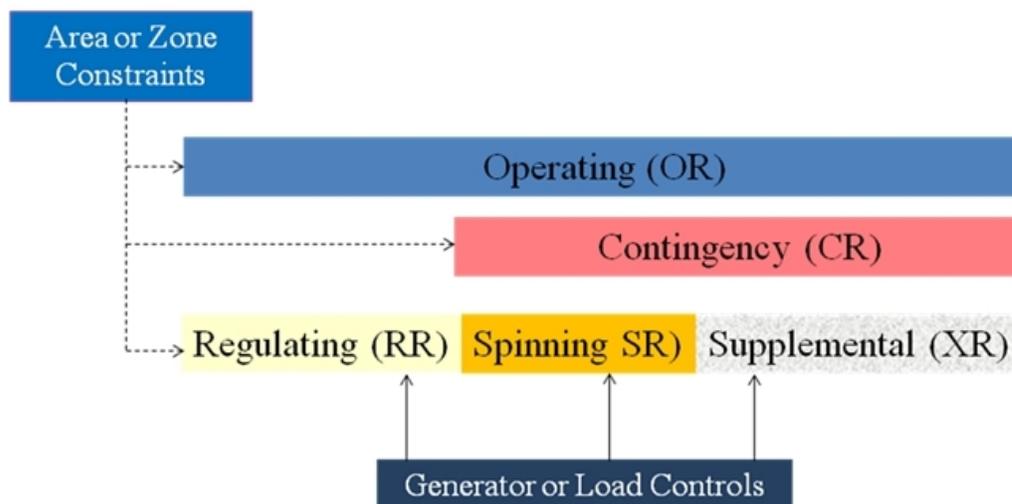
Contingency reserve plus regulating reserve up is called Operating Reserve (OR)

The following relations are then established:

$$RR^+ + SR + XR = OR$$

$$RR^+ + CR = OR$$

Data for the reserve controls can be specified from the [Generator Information dialog](#), the [Load Information dialog](#), or from the corresponding case information displays accessible from the **Model Explorer > Optimal Power Flow > Results > Reserve Results**.



Generator and Load OPF Reserves Bids

Generators and loads can both be providers of reserve services. Similar options exist for both in setting the OPF Reserve Bids parameters. These options can be set for generators on the [Cost tab](#) of the Generator Information dialog and for loads on the [OPF Load Dispatch tab](#) of the Load Information dialog.

OPF Regulating Reserves

These options are only available for generators.

Up Reserve MW - Value in MW of the cleared regulating reserve up. This value is determined by the OPF Reserves solution and is non-enterable by the user.

Down Reserve MW - Value in MW of the cleared regulating reserve down. This value is determined by the OPF Reserves solution and is non-enterable by the user. When a single bidirectional control is used for up and down reserve, the MW value of the cleared Up Reserve and Down Reserve will be the same.

Available for Regulating Reserves - Check this box to declare this unit as a provider of regulating reserves service.

Price [\$/MWh] - This is the bid submitted for the block of regulating up and down reserves.

Maximum MW Increase - This is the size in MW of the block of the regulating reserve up bid submitted to the market. The cleared regulating reserve up cannot be larger than this value.

Maximum MW Decrease - This is the size in MW of the block of regulating reserve down bid submitted to the market. The cleared regulating reserve down cannot be larger than this value.

OPF Contingency Reserves

These options are available for both generators and loads.

Spinning Available - Check this box to declare this unit/load as a provider of spinning reserve service.

Supplemental Available - Check this box to declare this unit/load as a provider of supplemental reserve service.

Reserve MW - Value in MW of the cleared spinning and supplemental reserves. These values are determined by the OPF solution and are non-enterable by the user.

Price [\$/MWh] - This is the bid submitted for the corresponding blocks of spinning and supplemental reserve.

Max MW Increase - For spinning and supplemental reserve this is the size of the reserves block submitted to the market. The cleared reserves cannot be larger than these values.

OPF Reserves Constraints

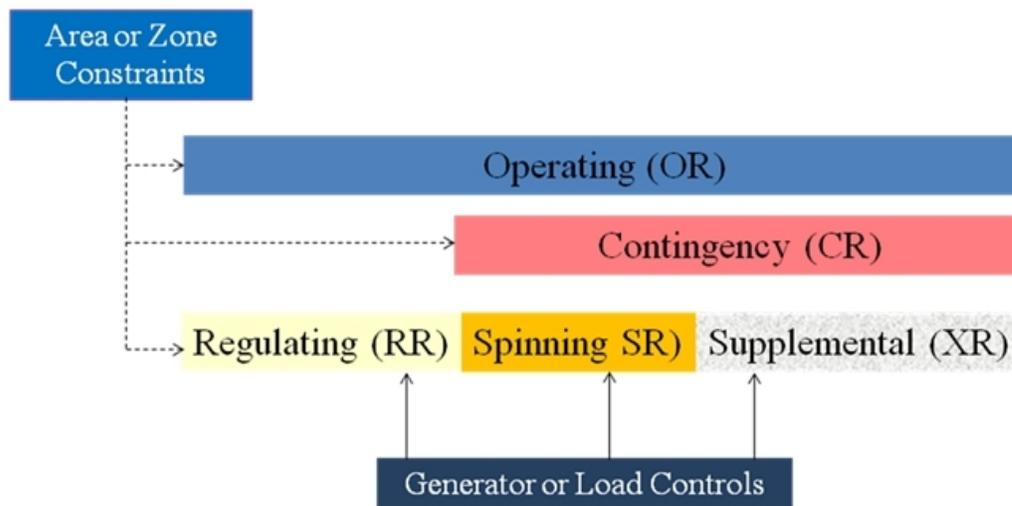
Reserve constraints in the OPF Reserves tool are specified at the area and zone level. Overlapping constraints are handled if an area contains several zones or if a zone spans several areas. For each type of reserve specified, the total reserve of the area or zone must be equal or greater than a certain level of reserve requirement in MW:

- Operating Reserve: $OR \geq OR_{REQ}$
- Regulating Reserve: $RR \geq RR_{REQ}$
- Contingency Reserve: $CR \geq CR_{REQ}$
- Some systems require that a percentage of the contingency reserve be spinning: $SR \geq \%CR$

Reserve requirements are input as incremental demand curves. These can be defined for both areas and zones and are described in more detail in the [Area and Zone Reserve Requirement Curves topic](#).

The reserve requirement demand curves are specified by pairs of (MW,\$/MWh) values that form a stair function. The demand curve should be monotonically decreasing, i.e., while the MW values increase, the \$/MWh values should decrease.

Sometimes, ancillary services markets rules specify a single reserve requirement value and price. For instance, 300MW at \$500/MWh. This requirement can be modeled by a demand curve that contains two points: 0.0 MW at 300 \$/MWh, and 300 MW at 0.0 \$/MWh.



Area and Zone OPF Reserve Requirement Curves

Simulator OPF Reserves requires the user to specify area or zone reserve constraints as reserve demand curves for different types of requirements: operating, regulating and contingency.

Simulator OPF Reserves allows reserve constraints to be specified at the area and zone level or both, including overlapping regions. The data needed to specify reserve constraints at the zone level is identical to that needed for area reserve constraints.

Reserves Requirement Curves are found on the OPF tab of the [Area Information dialog](#) and the OPF tab of either the [edit](#) or [run](#) mode Zone Information dialog.

The reserve requirement incremental demand curves are specified by a set of pairs of (MW,\$/MWh) values that form a stair function. The incremental demand curve should be monotonically decreasing, i.e., while MW values increase, \$/MWh values should decrease.

Sometimes, ancillary services markets rules specify a single reserve requirement and its price. For instance 300MW at \$500/MWh. This requirement would be modeled by a demand curve that contains two points: 0.0 MW at 300 \$/MWh, and 300 MW at 0.0 \$/MWh.

Enforce Operating Reserve Requirement in OPF - Check this box to make the specified operating reserve demand curve be enforced by the OPF Reserves tool.

Enforce Regulating Reserve Requirement in OPF - Check this box to make the specified regulating reserve demand curve be enforced by the OPF Reserves tool.

Enforce Contingency Reserve Requirement in OPF - Check this box to make the specified contingency reserve demand curve be enforced by the OPF Reserves tool.

Results Page - This page shows total results at the area level for different types of reserve quantities: regulating reserve up, regulating reserve down, spinning reserve, supplemental reserve, contingency reserve and operation reserve.

Enforce - Whether a specific type of reserve is enforced for the area. Only regulating, contingency and operation reserve can be enforce at the area or zone level.

Cleared MW - Total reserve cleared in the area, for each reserve type.

Max Reserve MW - Total available reserve in the area for each reserve type.

Hourly Cost \$/hr - Total cost of providing the reserve service in the area for each reserve type.

Price \$/MWh - Reserve Marginal Clearing Price (RMCP) for each reserve type. This price is a result of enforcing the reserve constraints, and thus it is only available for regulation, contingency and operation reserves.

Hourly Benefit \$/Hr - Benefit provided by enforcing reserve constraints at the area level. This benefit is derived from the enforced reserve demand requirements, and it is only available for regulation, contingency and operation reserves.

OPF Reserves Objective Function

The power system should operate exceeding level of the different types of reserves. The benefit associated with a reserve type is equal to the reserve consumer surplus (area below the reserve demand curve and above the equilibrium RMCP). In order to provide reserve, some generators will need to reduce their output to make room for the assigned reserve level. Thus in general, meeting the reserve requirements will result in increased energy prices. In Simulator, the OPF reserves problem is solved using an application of sequential linear programming embedded in the OPF solver. The LP problem consists in maximizing total surplus:

Total Surplus = Benefit - Costs

Where the benefits include:

- Up Regulation Reserve Surplus
- Down Regulation Reserve Surplus
- Contingency Reserve Surplus
- Operating Reserve Surplus

And the costs include:

- Up Regulation Reserve Cost
- Down Regulation Reserve Cost
- Spinning Reserve Cost
- Supplemental Reserve Cost

OPF Reserves Results and Pricing

The results of co-optimization of energy and reserves as implemented in Reserve Markets are:

- Optimal generation dispatch MW set-points and reserve assignments
- Energy locational marginal prices (LMP)
- Reserve market clearing prices (RMCP)

The energy locational marginal prices are different than the prices that would be obtained if reserve requirements are not consider. Usually, LMPs are higher in simultaneous energy and reserve optimization. This is due to some generators having to back-off their output to meet reserve control assignments.

As the bus LMP values correspond to the change in operating cost when an additional MW of active power load is served at that bus, the RMCP correspond to the change in operating surplus when an additional MW of a certain type of reserve is required to be provided at that bus. Each area or zone that specifies reserve requirements will have a RMCP. If particular, the RMCP of buses in overlapping zones and areas will be the result of adding the corresponding RMCP associated with each constraint. The bus components of the RMCP can be accessed from the [Bus MW Marginal Prices Details page](#) in the OPF Options and Results Dialog. RMCP fields will show up automatically for each constraint type that is binding.

OPF Reserves Case Information Display

A special category of [case information display](#) is available from the [Model Explorer](#) to access common OPF reserves data. From the Model Explorer select the **Optimal Power Flow > Results > Reserve Results**. The same set of reserve case information displays can be accessed from the [Add Ons ribbon tab](#) under the **Optimal Power Flow ribbon group > OPF Case Info > OPF Reserves**.

These case information displays include input and output data for the Reserves Markets, such as areas, zones, generator and loads. They have been customized with fields relevant to OPF Reserves. For instance, generator and loads include:

- Availability of each reserve type: regulating, spinning, supplemental
- Maximum reserve available for each reserve type
- Single value bid price for the reserve bid block, for each reserve type
- Resulting RMCP for each reserve type

The area and zone information displays include:

- Enforcement of reserve constraints: regulating, contingency, and operation
- Total cleared reserve for each type
- Total reserve cost and benefit for each type

Then, let us set up the following reserve blocks and prices (reserve bids) in the same case information diagram.

	REG Max Up	REG Max Down	SPN Max	SUP Max	REG Price	SPN Price	SUP Price
1	50.00	50.00	102.50	29.00	10.00	15.00	10.50
2	60.00	50.00	112.50	34.00	12.00	13.00	9.50
4	70.00	50.00	122.50	39.00	14.00	11.00	8.50
7	80.00	50.00	132.50	44.00	18.00	9.00	7.50
8	90.00	50.00	142.50	49.00	24.00	7.00	6.50
9	100.00	50.00	152.50	54.00	30.00	5.00	5.50
10	110.00	0.00	202.50	59.00	40.00	3.00	4.50

Once the generator and/or load reserve controls have been specified, the second step is to set up the reserve constraints for the control areas and/or zones. Let us set up the following data by accessing each individual [Area Information dialog](#) or [Zone Information dialog](#). Recall, that a single level reserve requirement such as 150 MW @ 400 \$/MWh is set by entering two points in the reserve demand curve: 0.0 MW at 400 \$/MWh and 150 MW at 0.0 \$/MWh.

	Enforce			Requirement		
	OPR	REG	CTG	OPR	REG	CTG
Area A	YES	YES		150 @ 400	80 @ 300	
Area B	YES	YES		90 @ 250	60 @ 150	
Zone 1			YES			65 @ 300
Zone 2			YES			45 @ 140
Zone 3			YES			35 @ 120

Let us now solve the OPF enforcing reserve requirements. Go to the [Add Ons ribbon tab](#) and choose [OPF Options and Results](#) in the Optimal Power Flow ribbon group. Check the box **Include OPF Reserve Requirements**. Click the Solve LP OPF button from the dialog or the Primal LP button from the Add Ons ribbon tab.

The OPF Reserves, will enforce reserve constraints for each reserve type and for each area and zone whose Enforce Field is set to *YES*. In addition, OPF Reserve will observe the following constraints:

- Transmission line limits
- Generator MW Max and Min limits for the total energy plus reserve controls
- Area scheduled interchange. For this case, there is a 100 MW export from Area A to Area B.

To explore the results, go to the Generators page in [OPF Reserve Results](#) in the Model Explorer. The results include generator MW dispatch and assignment of all the reserve services. Note that in this example, some of the generators have reached their reserve bid limits for certain units and types of reserve.

	Number of Bus	Name of Bus	ID	REG Avail.	SPN Avail.	SUP Avail.	Min MW	Min Reg MW	Gen MW	REG MW Up	REG MW Down	SPN MW Up	SUP MW Up	Max Reg MW
1	1	One	1	YES	NO	NO	50.00	0.00	160.00	50.00	50.00	0.00	0.00	250.00
2	2	Two	1	YES	YES	NO	50.00	0.00	80.00	30.00	30.00	65.00	0.00	300.00
3	4	Four	1	NO	YES	YES	50.00	0.00	74.72	0.00	0.00	0.00	5.00	200.00
4	7	Seven	1	YES	NO	YES	0.00	0.00	198.60	43.00	50.00	0.00	40.00	300.00
5	8	Eight	1	YES	YES	NO	150.00	0.00	191.68	0.00	10.00	0.00	0.00	300.00
6	9	Nine	1	NO	YES	YES	0.00	0.00	100.00	0.00	0.00	0.00	0.00	200.00
7	10	Ten	1	NO	NO	YES	0.00	0.00	99.87	0.00	0.00	0.00	7.00	1000.00

Results can be explored at the area or zone level by accessing the individual Area Information dialog or Zone Information dialog, going to the OPF tab, and the Results subtab of the [Reserve Requirement Curves](#). For instance, the Results for Area A are as shown in the next Figure. For each area, the results page shows the enforcement of different types of regulation, the total cleared reserve, the total available reserve, the reserve cost per hour, the reserve price, and the hourly benefit, which corresponds to the area below the demand curve and above the equilibrium RMCP level.

	REG +	REG--	SPN	SUP	CTG	OPR
Enforce	YES	YES	NO	NO	NO	YES
Cleared MW	80.00	80.00	65.00	5.00	70.00	150.00
Max Reserve MW	110.00	100.00	235.00	39.00	274.00	384.00
Hourly Cost \$/hr	860.00	860.00	845.00	42.50	887.50	1747.50
Price \$/MWh	143.00	12.00			-201.00	-131.00
Hourly Benefit \$/hr	24000.00	24000.00			0.00	60000.00

The Results tab of the [OPF Options and Results dialog](#) shows the details of Bus MW marginal prices, including energy, congestion and losses, and the reserve prices associated with the area or zone reserve constraints. The corresponding linear programming details will list among the LP variables the reserve controls specified by the user. The LP tableau will include specific rows for the enforced reserve constraints at the area and zone level, etc. Reserve results can be visualized as any other generator, load or bus object field in Simulator.

Available Transfer Capability (ATC) Analysis

The ATC Analysis tool is only available if you have purchased the ATC add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the ATC version of Simulator.

Available Transfer Capability (ATC) analysis determines the maximum incremental MW transfer possible between two parts of a power system without violating any specified limits. Most often, this transfer is between two areas in the system, but can be customized to specific groups of system devices.

Simulator's ATC analysis makes use of several tools that are available elsewhere in Simulator. These include:

- [Power Transfer Distribution Factors \(PTDFs\)](#): determine the linear impact of a transfer (or changes in power injection) on the elements of the power system.
- [Line Outage Distribution Factors \(LODFs\)](#): determine the linear impact of a line outage on the elements of the power system.
- [Contingency Analysis](#): studies the impact of a list of contingencies on the power system.
- [Limit Monitoring Settings](#): control which elements of the system are monitored for limit violations.

You do not directly use these other tools when using Simulator's ATC analysis tool, but Simulator uses the settings and algorithms in the background to determine ATC. Thus, it is helpful to be knowledgeable on their use, as it will help you in interpreting the results of an ATC analysis.

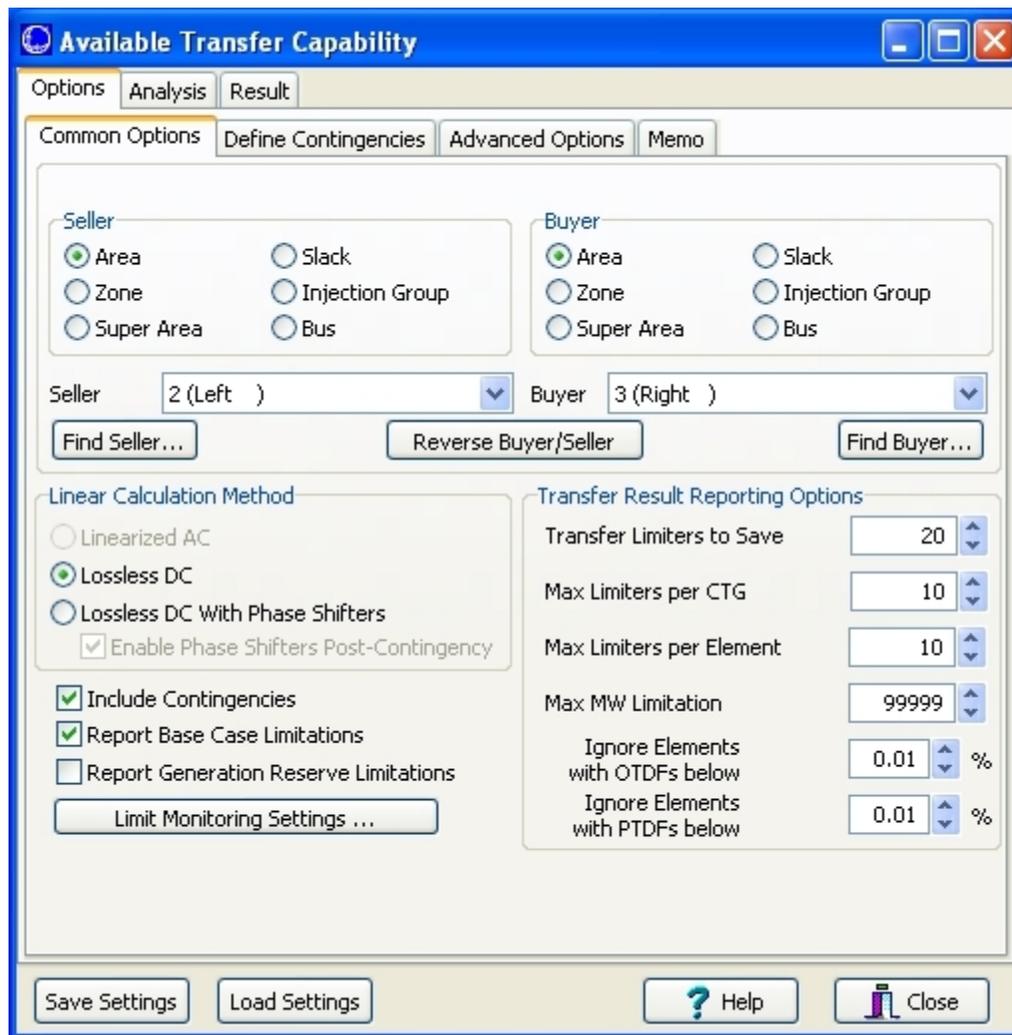
- Simulator provides three methods of determining the ATC for a transfer direction. See : [ATC Solution Methods](#)

For information on how to use the Simulator ATC tool, see [Available Transfer Capability Dialog](#).

Available Transfer Capability Dialog

The Available Transfer Capability Dialog provides an interface for performing and viewing the results of [Available Transfer Capability Analysis](#). ATC analysis is typically only done on the present power system state or scenario (called Single Scenario ATC Analysis). Click [here](#) for information on performing [multiple scenarios](#) analysis.

To open the ATC Dialog, go to the [Add Ons](#) ribbon tab and select **Available Transfer Capability (ATC)** from the **ATC** ribbon tab. The dialog opens to the Single Scenario ATC Analysis version with the Options tab, Common Options sub-tab visible.



The ATC Dialog is divided in three tabs: [Options tab](#), [Analysis tab](#), and [Result tab](#). If **Analyze Multiple Scenarios** has been checked on the [Advanced Options sub-tab](#) of the [Options tab](#), the layout of the ATC Dialog is slightly different—see [Multiple Scenarios ATC Dialog](#).

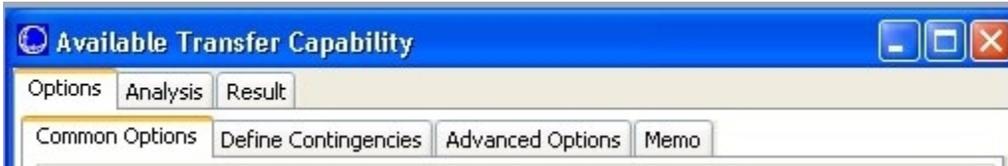
In addition, the following buttons can be used to save/load ATC analysis settings:

Save/Load Settings

ATC Analysis settings can be saved by selecting **Save Settings** on the ATC dialog. This allows you to repeat the analysis without having to reconfigure the settings. Select **Load Settings** to retrieve previously saved settings.

ATC Dialog: Options

The Options tab is available on the [Available Transfer Capability dialog](#). This tab is subdivided into four sub-tabs:

**Common Options****Define Contingencies**

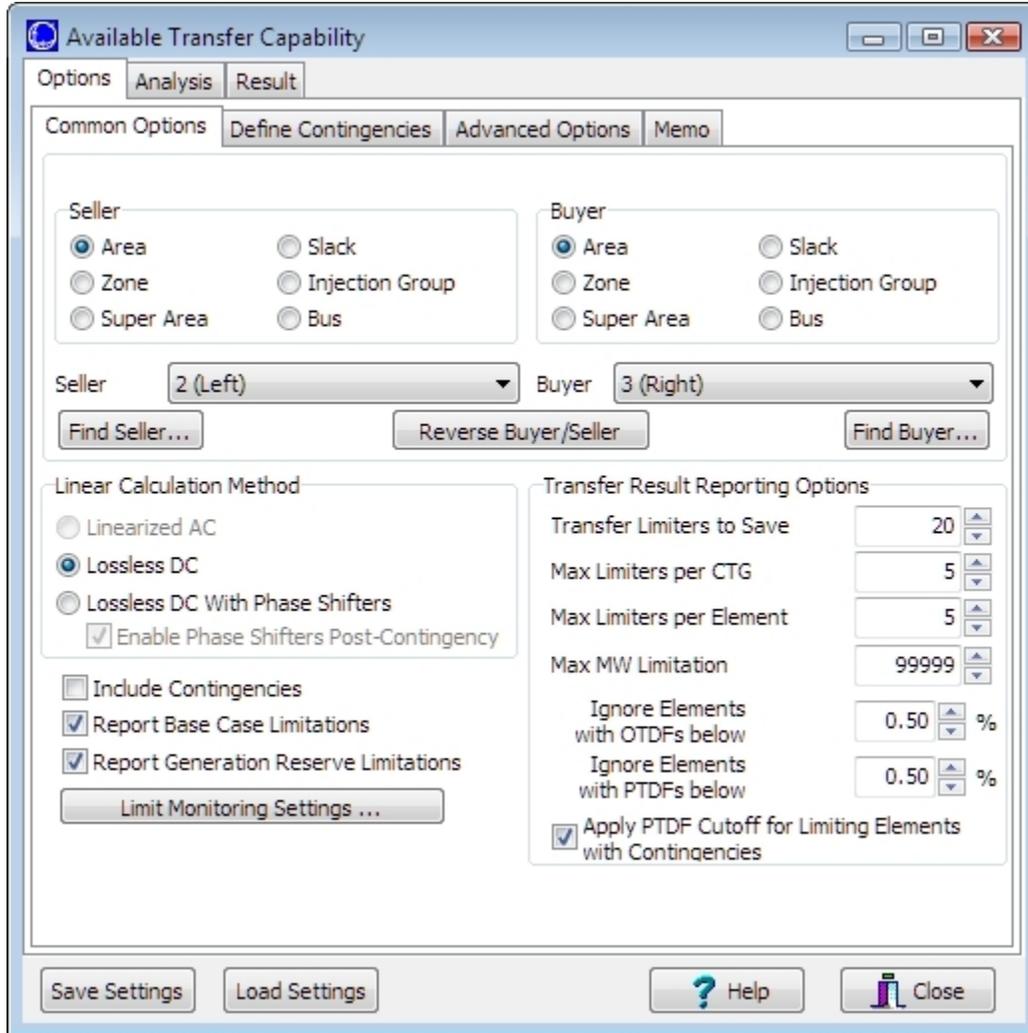
This sub-tab is similar to the [Contingencies Tab](#) of the [Contingency Analysis Dialog](#). The user can insert, auto-insert, define and/or delete contingency records. See [Contingency Analysis](#) for detailed information on defining contingencies.

Advanced Options**Memo**

Notes related to the ATC analysis, or anything else for that matter, can be entered on this sub-tab. These notes will be saved to an auxiliary file if choosing to save the ATC settings.

ATC Dialog Options: Common Options

The Common Options sub-tab is found on the [Options](#) tab of the [Available Transfer Capability dialog](#).



The following parameters can be set on the Common Options sub-tab:

Seller Type, Buyer Type

For the [ATC Solution Method of Single Linear Step \(SL\)](#), ATC can be calculated for transfers between combinations of areas, zones, [super areas](#), [injection groups](#), buses, or to the system slack bus. For the iterated linear step ATC Solution Methods ([IL](#) and [IL then full contingency solution](#)), ATC can be calculated for transfers between areas, [super areas](#) or [injection groups](#) only. Use the seller type and buyer type options to indicate the type of the selling and purchasing entities.

Seller, Buyer

These dropdown boxes allow you to select the selling and buying entities. The contents of each depend on the Seller/Buyer Types selected. Clicking the **Find Seller** and **Find Buyer** buttons allows you to use Simulator's [Advanced Search Engine](#) to locate the desired entities.

The Seller defines where power is injection into the system and the Buyer defines where power is taken out of the system. The transfer is simulated by increasing generation or decreasing load in the Seller and decreasing generation or increasing load in the Buyer. The [ATC Analysis Methods - Solution Methods](#) go into greater detail about how the impact of the transfer is actually determined. When areas, zones, or super areas are selected to be either the Seller or Buyer, only on-line generators in the defined group that are on AGC control are allowed to participate in the transfer. Each generator participates in the transfer in proportion to its participation factor. When injection groups are selected to be either the Seller or Buyer, only on-line generators and loads defined with the injection group are allowed to participate in the transfer. Each element participates in the transfer in proportion to the participation factor defined with its [participation point](#) in the injection group. When a single bus is selected as the Seller or Buyer then all of the power injection change needed for the transfer comes from that bus. It does not matter if there is a generator or load at that bus. When the slack is selected for the Seller, then all of the power injection change needed for the seller side of the transfer comes from the island slack bus for the island in which the Buyer belongs. When the slack is selected as the Buyer, then all of the power injection change needed for the buyer side of the transfer comes from the island slack bus for the island in which the Seller belongs.

Generator and load MW limits are not enforced regardless of the type of buyer and seller. Error messages will be displayed if the Seller and Buyer are not both completely contained within the same electrical island.

Reverse Buyer/Seller Button

Click this button to reverse the direction currently shown. The buyer becomes the seller, and the seller the buyer.

Linear Calculation Method

The ATC analysis tool can use either a **Lossless DC** or **Lossless DC with Phase Shifters** calculation method for obtaining the ATC results.

If you select the Lossless DC option, branch flow sensitivity is calculated by estimating the real power that flows through the monitored element only from the difference in angles measured across its terminals.

The Lossless DC with Phase Shifters method, a modification to the lossless dc approximation, takes into account phase shifter operation. It is especially useful when the ATC tool continually reports overloads on branches that obviously will not overload because of the operation of a phase shifting transformer. This method assumes that the phase shifter angles may change to any value to ensure that the line flow on those lines does not change.

The Linearized AC method is not yet available.

Enable Phase Shifters Post-Contingency

This option becomes enabled when you choose the **Lossless DC with Phase Shifters** linear calculation method. It allows you to choose whether or not phase shifter control should be enforced during post-contingency ATC calculations. When this option is checked, it is assumed that phase shifter angles may change to keep the line flow from changing after the contingency has been applied. Otherwise, it is assumed that phase shifters only maintain the flow in the pre-contingency state and the angles do not change in the post-contingency state. Base case (pre-contingency) transfer limiters will still be determined with phase shifters enforced, regardless of the setting of this option when the Lossless DC with Phase Shifters option is checked.

Include Contingencies Check Box

Check this box to include contingencies (inserted or loaded using the [Contingency Analysis Tool](#)) in the ATC analysis. Note that minimizing the number of contingencies considered greatly improves solution speed as well as computer memory requirements for doing ATC Analysis. Therefore, be

careful in choosing which contingencies to use with the ATC tool. Setting the **Skip** field to *YES* for a particular contingency will exclude it from the ATC analysis.

The ATC tool uses the same contingency records as the contingency analysis tool. Contingency results may be invalid following ATC analysis. If necessary, save contingency results to an auxiliary file prior to running the ATC analysis.

Contingency elements have an associated [Status](#) that determines under what conditions a particular action will be implemented. When ATC values are calculated using linear methods, contingency actions are not actually implemented. Because contingency actions are not actually implemented, *POSTCHECK* actions cannot be applied after other actions have been performed and the power flow has been solved. Instead, *POSTCHECK* actions act as *CHECK* actions during [linear ATC analysis \(Single Linear Step Method\)](#). During linear ATC analysis whether or not model criteria are met for *CHECK* and *POSTCHECK* contingency actions will be determined at the base transfer level rather than at the transfer level that determines the limiting transfer. The base transfer level is the transfer level at which the linear ATC values are actually determined. In most cases this will be the zero transfer level, but this can be different if using one of the iterated ATC methods.

Report Base Case Limitations Check Box

When checked, the ATC tool will report transfer limitations from the base case (pre-contingency).

Report Generation Reserve Limitations Check Box

When checked, the ATC tool will report transfer limitations from generation reserve.

Generator and load MW limits are not enforced when calculating linear sensitivities used to determine the transfer MW limits. However, checking this option provides an indication of when power transfers are limited based on generator maximum or minimum MW limits or the amount of load that is on-line. When this option is checked and there are limitations to how much power can be transferred because of max/min generator limits or the on-line load, additional Transfer Limiters will be present in the [results](#). The Limiting Element will indicate either the Buyer or Seller and the Transfer Limit in MW will be the amount of transfer that can be achieved before the Buyer or Seller exceeds some limitation to generation or load. When considering reserve limits for the Seller, generators will be included in the reserve limit by taking the difference between the generator maximum MW limit and the present output and loads will be included in the reserve limit by the present amount of on-line load. When considering reserve limits for the Buyer, generators will be included in the reserve limit by taking the difference between the present output of the generators and the minimum MW limit and loads will be included by an arbitrary value of 9999.9 MW. Loads are included in this manner because maximum and minimum limits on loads are not currently used in the ATC tool.

For example, assume than an area that currently has 200 MW on-line and a total maximum MW output of 250 MW for all on-line generators that are on AGC control is defined as the Seller. A 50 MW generation reserve limitation will be reported for this area. As another example, assume that an injection group currently has 50 MW of on-line generation with a total maximum MW output of 150 MW and 50 MW of on-line load is defined as the Seller. A 150 MW reserve limitation will be reported for this injection group.

No generation reserve limitations are reported when the seller or buyer type is a bus or the slack.

If using one of the iterated methods, this option will be ignored except during the first single linear step. Reserve limitations may be reported from this step, but limitations of this type will not be iterated on during the individual iteration process. If choosing to enforce generator MW limits appropriately with the type of transactors that are defined, reserve limits will still be reported, regardless of this option setting, if they are hit during any step of the iterative process when ramping is occurring. See the detailed descriptions of either the [Iterated Linear Step \(IL\)](#) or [Iterated Linear Step \(IL\) then Full CTG Solution](#) method for more information about how reserve limits are checked during the iterated methods.

Limit Monitoring Settings Button

Click this button to open the [Limit Monitoring Settings Dialog](#). Limiting Elements will only be reported for those elements that are selected to be monitored. Note that minimizing the number of monitored power system elements greatly improves solution speed as well as computer memory requirements for doing ATC Analysis.

[Monitoring Exceptions](#) can be used to include or exclude specific monitored elements on a contingency-by-contingency basis.

Transfer Result Reporting Options

The options in this section of the ATC dialog are discussed in detail in the topic on [Transfer Result Reporting Options](#).

ATC Dialog Common Options: Transfer Result Reporting Options

These options are found in the Transfer Result Report Options section on the [Common Options](#) sub-tab of the [Options](#) tab found on the [Available Transfer Capability dialog](#).

Transfer Limiters To Save

This value tells Simulator how many total Transfer Limiters to save. Simulator will save those Transfer Limiters with the lowest Transfer Limitation. An explanation of a Transfer Limiter follows:

During Linear ATC Analysis, Simulator determines the Transfer Limitation (See [Available Transfer Capability Analysis](#)) for each transmission line and interface during each contingency and the base case. From this Simulator develops a list of Transfer Limiters. A Transfer Limit contains three key pieces of information:

- Transfer Limit in MW
- Limiting Element: Transmission branch (or interface) that causes the limit
- Limiting Contingency: Contingency that is applied to cause the Limiting Element to overload (if it is a limit without any contingency applied, then the contingency will say *Base Case*)

Thus if we are monitoring 1000 transmission lines during 99 contingencies plus 1 base case, there would be 100,000 Transfer Limiters calculated. We are not concerned with all 100,000 limitations, therefore, only the limitations with the smallest Transfer Limit in MW are reported.

Max Limiters per CTG

When analyzing a long list of contingencies, the worst transfer limitations may all occur during the same contingency. Set this value to limit the number of Transfer Limiters saved that are associated with one contingency.

Max Limiters per Element

When analyzing a long list of contingencies, the worst transfer limitations may all be overloads of the same limiting element. Set this value to limit the number of Transfer Limiters saved that are associated with one Limiting Element.

Max MW Limitation

This value defines the maximum transfer to report between the buyer and seller. Simulator will compute the ATC analysis results until the transfer amount reaches the value in this field and only report the results meeting the other reporting criteria up to this MW limitation.

When using one of the iterated methods, this option is only applied during the initial Single Linear Step portion of the method. Once individual limiters are being iterated on, this option will be ignored.

Ignore Elements with OTDFs below

Simulator will not report Transfer Limitations for elements with [OTDF](#) values (or [PTDF](#) values if there is no Limiting Contingency) less than this user-specified value. The default value is 0.5%, meaning that for a 100 MW transfer, there would be only a 0.5 MW increase in flow on the Limiting Element.

The transfer limitation functions involve dividing by the [PTDF](#) or [OTDF](#) values for each branch or interface to calculate the Transfer Limit. This leads to two facts:

- The accuracy of the transfer limitation is less for lines that have very small [PTDF](#) or [OTDF](#) values.
- A very small [PTDF](#) or [OTDF](#) value means that the transfer has very little impact on the line.

These two facts often result in Linear ATC analysis reporting inaccurate transfer limitations for lines that are largely unaffected by the transfer. It is not uncommon to have a transfer limitation report an extremely negative transfer limit (e.g. $-1.9E28$ MW). A branch which is overloaded by a very small percentage, but which has a very small OTDF value often causes this. If the OTDF value is 0.001%, then a branch overloaded by 1 MW will result in a transfer limitation of -100,000 MW.

This motivates the usefulness of ignoring elements with small [PTDFs](#) and [OTDFs](#).

Ignore Elements with PTDFs below

Simulator will not report Transfer Limitations for elements with [PTDF](#) values less than this user-specified value. The default value is 0.5%.

Apply PTDF Cutoff for Limiting Elements with Contingencies

When this option is checked, for an element that is being monitored for a contingency condition, the OTDF for the monitored element with the contingency in place must meet the OTDF cutoff, and the PTDF for the monitored element without the contingency in place must meet the PTDF cutoff.

If this option is not checked, only the OTDF cutoff will be considered when evaluating a monitored element with a contingency applied. There will be no check for the same monitored element without the contingency applied using the PTDF cutoff.

Regardless of how this option is set, the PTDF cutoff will only be considered when evaluating a monitored element in the base case.

The cutoffs applied to interfaces are also independent of how this option is set. For an interface that contains contingency elements and is being monitored in the base case, the minimum of the PTDF and OTDF cutoffs will be used. For an interface that does not contain any contingency elements and is being monitored in the base case, the PTDF cutoff will be used. Interfaces that contain contingency elements cannot have additional contingencies applied on top of them in the ATC analysis. Interfaces that do not have contingency elements can have contingencies applied on top of them. The OTDF cutoff will be used when evaluating these types of interfaces with contingencies in place.

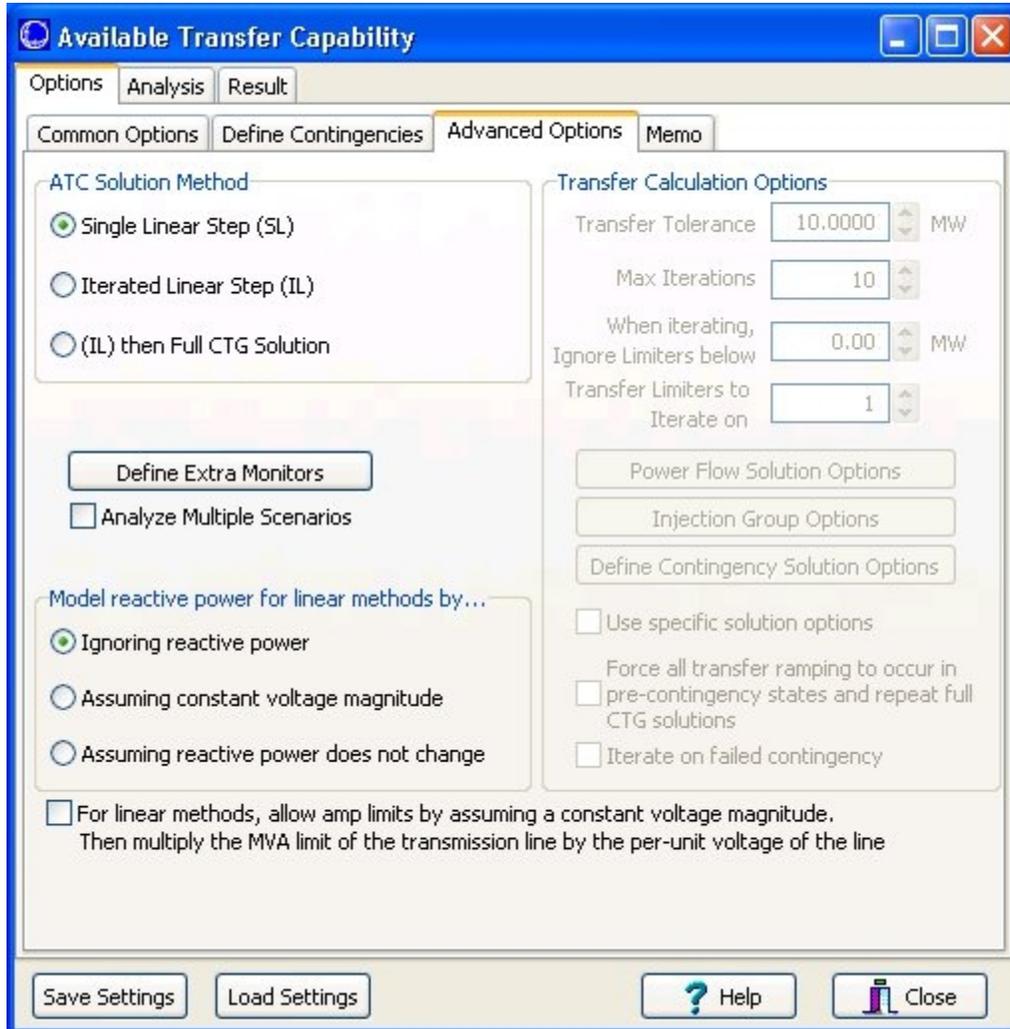
All of these options work together to determine the list of limiters that shows up in the results. The [Limit Monitoring Settings](#) determine elements that will be considered for monitoring. The [list of defined contingencies](#) determines the contingencies. Every combination of monitored elements and contingencies will be tested. [Monitoring exceptions](#) can also be used to include or exclude monitored elements on a contingency-by-contingency basis. If the PTDF and/or OTDF cutoffs, as appropriate, are met for the monitored element/contingency pair, the monitored element and contingency are then checked against the **Max Limiter per Element** and **Max Limiters per CTG** conditions.

For each monitored element, a maximum, X, of **Max Limiter per Element** is kept during the initial processing. Only the most limiting X limiters for that element are retained. The transfer limiter with the highest transfer limit value will get pushed out of the list for an element if a new combination of monitored element/contingency is more limiting than what is currently in the list. For each contingency, a maximum, Y, of **Max Limiters per CTG** is kept during the initial processing. Only the most limiting Y limiters that for contingency are retained. The transfer limiter with the highest transfer limit value will get pushed out of the list for a contingency if a new combination of monitored element/contingency is more limiting than what is currently in the list. Once all monitored element/contingency pairs have been processed, the intersection of the element lists and the contingency lists is found. This means that the same limiter must show up in the top X limiters for its element list and in the top Y limiters for its contingency list for it to be kept for the final results. Once the element and contingency lists are merged, the final list of limiters to report is limited by the number of **Transfer Limiters to Save** that are below the **Max MW Limitation**. If there are no monitored element/contingency pairs that are below the Max MW Limitation, the monitored

element/contingency pair with the lowest transfer limit value will be the only record reported in the results.

ATC Dialog Options: Advanced Options

The Advanced Options sub-tab is found on the [Options](#) tab of the [Available Transfer Capability dialog](#).



The following parameters can be set on the Advanced Options sub-tab:

ATC Solution Method

One of the [solution methods](#) for determining the ATC of a specified transfer direction.

Define Extra Monitors

Simulator's ATC tool determines the maximum amount of MW transfer between the seller and buyer. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, you can utilize Extra Monitors. Click the **Define Extra Monitors** button to open the [ATC Extra Monitors Dialog](#). The flow on the extra monitors will be reported for each transfer limiter determined.

Analyze Multiple Scenarios

Check Analyze Multiple Scenarios to perform ATC Analysis on several scenarios. See [ATC Dialog for Multiple Scenarios](#) for more information on Multiple Scenario Analysis.

Model Reactive Power for Linear Methods By...

The linearized methods used in the ATC tool are based only on the changes in real power MW in the system, thus an assumption needs to be made about the reaction of the Mvar flows during the linear calculations. The chosen assumption allows a MW limit to be calculated from the defined MVA limit for use in determining the ATC values. Before applying the option for handling reactive power, the specified MVA limit for a branch is used to determine the starting MVA limit. The [Limit Monitoring Settings](#) can cause an adjustment the defined MVA limit based on the **Branch Percentage** specified. The modified limit that is used in the calculations is then:

$$MVALimit = DefinedMVALimit * BranchPercentage$$

The choices for adjusting the MVA Limit for reactive power are then:

Ignoring reactive power – reactive power is completely ignored. The MW limit of a line is assumed to be the modified MVA limit of the line:

$$MWLimit = MVALimit$$

Assuming constant voltage magnitude – reactive power is adjusted to hold voltage magnitudes constant. This option uses the intersection of a branch's operating circle and limiting circle to adjust the limit. The operating circle defines a circle of valid MW and Mvar values for a branch as a transfer takes place across the system. The limiting circle has a radius equal to the modified MVA limit of the branch.

Assuming reactive power does not change – reactive power is held at pre-ATC solution levels. The MW limit for each line is calculated from the MVA limit and the reactive power at the pre-ATC solution level:

$$MWLimit = \sqrt{(MVALimit)^2 - (Mvar)^2}$$

For Linear Methods, Allow Amp Limits by Assuming a Constant Voltage Magnitude

If checked, Simulator will allow converting MVA limits to Amp limits by assuming constant voltage magnitudes based on the base case full AC load flow operating point just prior to the ATC linear calculations. This option will only be used if the [Limit Monitoring Settings Option of Treat Line Limits as Equivalent Amps](#) is checked.

This option will cause the ATC tool to internally use a modified MVA limit which is equal to the user-specified MVA rating multiplied by the actual terminal voltages of the line. This results in a modified MVA rating which is higher than the user-specified MVA rating when the voltage on the line is higher than nominal and lower when the voltage is lower than nominal. For purposes of calculating the ATC values, a MW limit is determined from the MVA rating based on how this option is set and how the reactive power is modeled.

Ignoring reactive power

$$MWLimit = MVALimit * V$$

Assuming reactive power does not change

$$MWLimit = \sqrt{(MVALimit * V)^2 - (Mvar)^2}$$

where V is the voltage at the end of the branch from which the limit is taken. The Transfer Limit is calculated at each end of a branch and the most limiting value is reported as the Transfer Limit for a given branch.

Transfer Calculation Options

Note: The Transfer Calculation Options section is disabled if the [Single Linear Step](#) Solution Method is selected.

Transfer Tolerance

This is the tolerance used during the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The iterated methods will finish if the transfer step amount for the next iteration is less than this tolerance. The default is 10 MW.

Max. Iterations

This is the maximum number of iterations used in the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The default is 10.

When Iterating, Ignore Limiters below

The iterated techniques perform additional analysis on individual transfer limitations in the order of increasing Transfer Limit in MW. When using one of the iterated techniques, transfer limitations with Transfer Limit values below a certain threshold can be ignored for the iterated portion of the process. Transfer limitations with Transfer Limit values equal to or below the value set with this option may still appear in the results, but they will not be iterated on individually.

Once an individual limiter is being iterated on, this value will no longer be used. This means that during the individual iteration process, the transfer level for a limiter can fall below this threshold.

Transfer Limiters to Iterate on

This is the number of transfer limiters to iterate on in the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The default is 1. The transfer limiters actually iterated on are determined by selecting the number of limiters set in this option in ascending order of Transfer Limit with the Transfer Limit at or above the value set in the **When Iterating, Ignore Limiters below** option.

Power Flow Solution Options

This button brings up the [Power Flow Solution Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases and the options used when no contingency-specific solution options are defined. Changing these options affects all power flow computations, even those outside the ATC process.

Because the goal is to stress the system while performing the ATC analysis, there are two power flow solution options that are set internally as part of the ATC process. These are the options that specify the minimum per unit voltage for constant power and constant current loads. These options are both set to 0 pu and override any user-specified settings.

Minimum Per Unit Voltage for	
Constant Power Loads	0.000
Constant Current Loads	0.000

These options are both set to 0 pu during both the base case solution and any contingency case solution.

Injection Group Options

This button brings up a dialog from which options associated with injection groups can be set. These are the same options that get set for injection groups when using [Island-based AGC](#) and dispatching using an injection group; the ATC process switches to using an island-based AGC when iterating and using injection groups for the seller and buyer. The options that can be set

are Island AGC Tolerance, Allow only AGC units to vary, Enforce unit MW limits, Do not allow negative loads, and how reactive power load should change as real power load is changed.

The sink injection group is set as the injection group to use in the dispatch. When implementing the transfer ramping, the source injection group is ramped to the required amount and then the power flow is solved with the sink injection group picking up any required changes. These changes include adjusting injection in response to change in the source injection group as well as any changes in losses due to the transfer.

Define Contingency Solution Options

Click this button to open the [Contingency Solution Options Dialog](#). These are the same options that are set with the contingency analysis options.

Use Specific Solution Options

When checked, the ATC tool will use the solution options defined by pressing the **Define Contingency Solution Options** button for contingency analysis. When not checked, all solutions will use the options defined by pressing the **Power Flow Solution Options** button.

Regardless of the specific contingency solution option settings, there are two options that will be set internally by the ATC process and will override any user-specified settings. These are the options for minimum per unit voltage for constant power and constant current loads. These options are described in more detail in the Power Flow Solution Options section above.

Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions

When using the iterated solution methods, the typical iterated operation is to solve for the transfer step amount in the post-contingency state, and then apply the transfer step amount to the post-contingency state and resolve for the next transfer step amount. When this option is checked, the transfer step amount for each iteration is calculated in the post-contingency state, but the power flow state is restored to the pre-contingency state before the transfer step amount is applied. The contingency is then applied after the transfer step amount has been ramped and the transfer step amount for the next iteration is calculated after the contingency has been applied. The purpose of this is to take into account the possibility of conditional actions in the contingency definitions that may not be active in the early steps of the iterated calculations, but may become active at some point during the application of the iterated steps.

Iterate on failed contingency

This option is only enabled when the option to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions** is selected. When this option is checked and the power flow fails to solve after the contingency has been applied, a new iteration loop is entered to determine the highest transfer level at which the contingency can be solved. This loop attempts to determine a more accurate transfer level at which the contingency fails to solve rather than simply reporting the transfer level at which the last power flow was successful regardless of the current step size. To start the iterative process, the current step size is reduced by half. The ramping is performed and the contingency is solved. The step size will remain the same as long as the contingency can be solved. If the contingency fails to solve, the step size will be reduced by half again. The ramping will continue until the contingency no longer solves, the accumulated transfer during the process meets the original step size, or the step size becomes smaller than the step size tolerance.

An additional check is done during the iterative process on the failed contingency. When the contingency does solve, the flow on the monitored element is checked to determine if it exceeds its limit. If it is overloaded, the step size will be reduced by half until the monitored element is no longer over its limit. While this check is being done on the monitored element limit, the ramping continues out to the specified amount and the contingency is solved.

During the entire iterative process on a failed contingency solution, the step size is limited to be between zero and the step size which was ramped when the contingency failed to solve for the first time.

ATC Extra Monitors Dialog

Simulator's ATC tool determines the maximum amount of MW transfer between the seller and buyer. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, you can utilize ATC Extra Monitors. To open the Extra Monitors dialog, click the **Define Extra Monitors** button on the [Advanced Options](#) tab of the ATC Analysis Dialog.

The ATC Extra Monitors Display lists all of the ATC Extra Monitors defined. This list display is a [Case Information Display](#), and therefore has the same functionality as other common displays. See **Adding and Removing Extra Monitors** below.

ATC ExMon Desc	ATC ExMon Obj	Monitor Limit	Relative Monitor Sensitivity Constant
1 MW flow	Branch Three (3) TO Four (4) CKT 1	100.00	none

Extra Monitors Dialog

The default fields shown on this display are:

ATC ExMon Desc

This is a description of the monitored value. Presently, this is always MW flow.

ATC ExMon Obj

The power system element that is being monitored. This will be either a transmission branch or an interface.

Monitor Limit

This is the MW limit of the element being monitored.

Relative Monitor

Set this to a positive value to further filter the Transfer Limitations reported on the [Transfer Limiters Display](#). By default, Relative Monitor is set to *none*, and no additional filtering of limitations is performed. If this value is greater than zero, only Transfer Limitations that meet the following condition are included in the results.

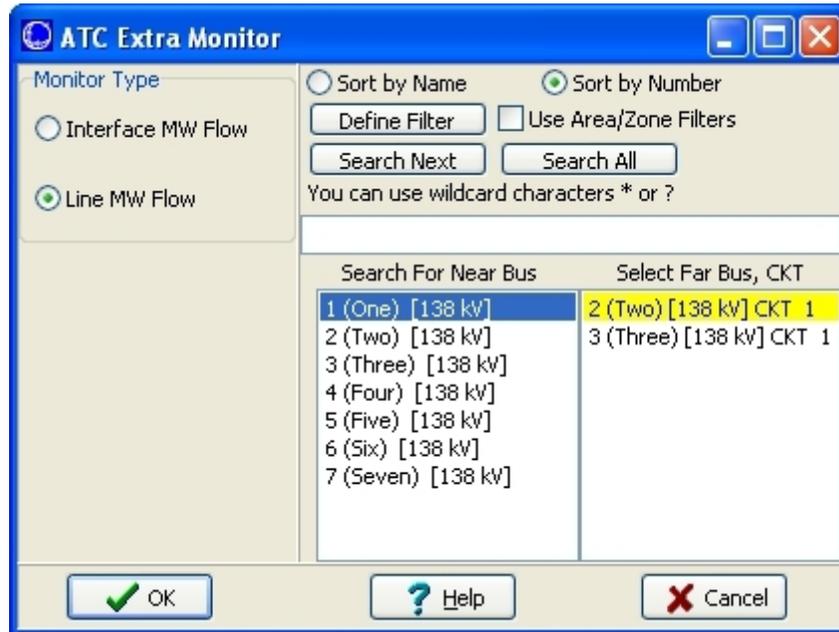
$$\frac{\left(\frac{OTDF_{M,C}}{LimitUsed_M} \right)}{\left(\frac{PTDF_{ExtraMon}}{Rating_{ExtraMon}} \right)} \geq (Relative\ Monitor\ Sensitivity\ Constant)$$

This provides a measure of how much an interface or branch is affected by the transfer relative to its MW limit.

The relative monitor sensitivity check is not done if using one of the [iterated ATC methods](#) and in the process of iterating on individual limiters. The assumption is that once a limiter meets the initial check and it is being iterated on that the limiter should be kept regardless of its relative monitor sensitivity.

Adding and Removing Extra Monitors

To delete an Extra Monitor, right-click on the desired record on the list display and select **Delete**. To insert an Extra Monitor, right-click on the list display and select **Insert**. This opens the insert dialog.



Insert Extra Monitors Dialog

Choose whether you want to monitor an Interface or Line MW flow. Next choose the interface or branch to be monitored. Note that the insert dialog allows the use of Simulator's [Advanced Search Engine](#) and techniques to aid in locating the desired interface or branch. Click **OK** to insert the record.

ATC Dialog: Result

The Result tab is found on the [Available Transfer Capability dialog](#). This tab is visible when performing ATC analysis on a single system state. The Result tab consists of two sections: A [Transfer Limiters Display](#) and a [Contingency Definition Display](#).

The screenshot shows the 'Available Transfer Capability' dialog box with the 'Result' tab selected. The dialog has three main sections: 'Options', 'Analysis', and 'Result'. Under 'Result', there are four sub-tabs: 'All Limiters', 'Branch Limiters', 'Interface Limiters', and 'Nomogram Interface Limiters'. The 'All Limiters' sub-tab is active, displaying a table with the following data:

	Trans Lim	Limiting Element	Limiting CTG	% OTDF	Pre-T Est
1	33.89	Branch Two (2) TO Five (5) CIL_00006Six-00007SevenC1		39.01	9%
2	33.89	Branch Two (2) TO Five (5) CIL_00006Six-00007SevenC2		39.01	9%
3	34.36	Branch Two (2) TO Five (5) CIL_00004Four-00005FiveC1		33.33	9%
4	52.97	Branch Two (2) TO Five (5) CIL_00002Two-00004FourC1		27.57	9%
5	54.36	Branch Four (4) TO Five (5) CL_00002Two-00005FiveC1		21.45	5%
6	55.46	Branch Two (2) TO Five (5) CIL_00002Two-00003ThreeC1		27.22	9%
7	201.32	Area Right (3)	Base Case	0.00	(
8	290.23	Branch Four (4) TO Five (5) CL_00006Six-00007SevenC1		14.65	2%
9	290.23	Branch Four (4) TO Five (5) CL_00006Six-00007SevenC2		14.65	2%
10	299.66	Area Left (2)	Base Case	0.00	(
11	316.25	Branch One (1) TO Three (3) L_00002Two-00003ThreeC1		3.23	5%

Below the table is the 'Contingency Definition' section, which contains a table with the following data:

	Actions	Model Criter
1	OPEN Branch Six_138.0 (6) TO Seven_138.0 (7) CKT 1	

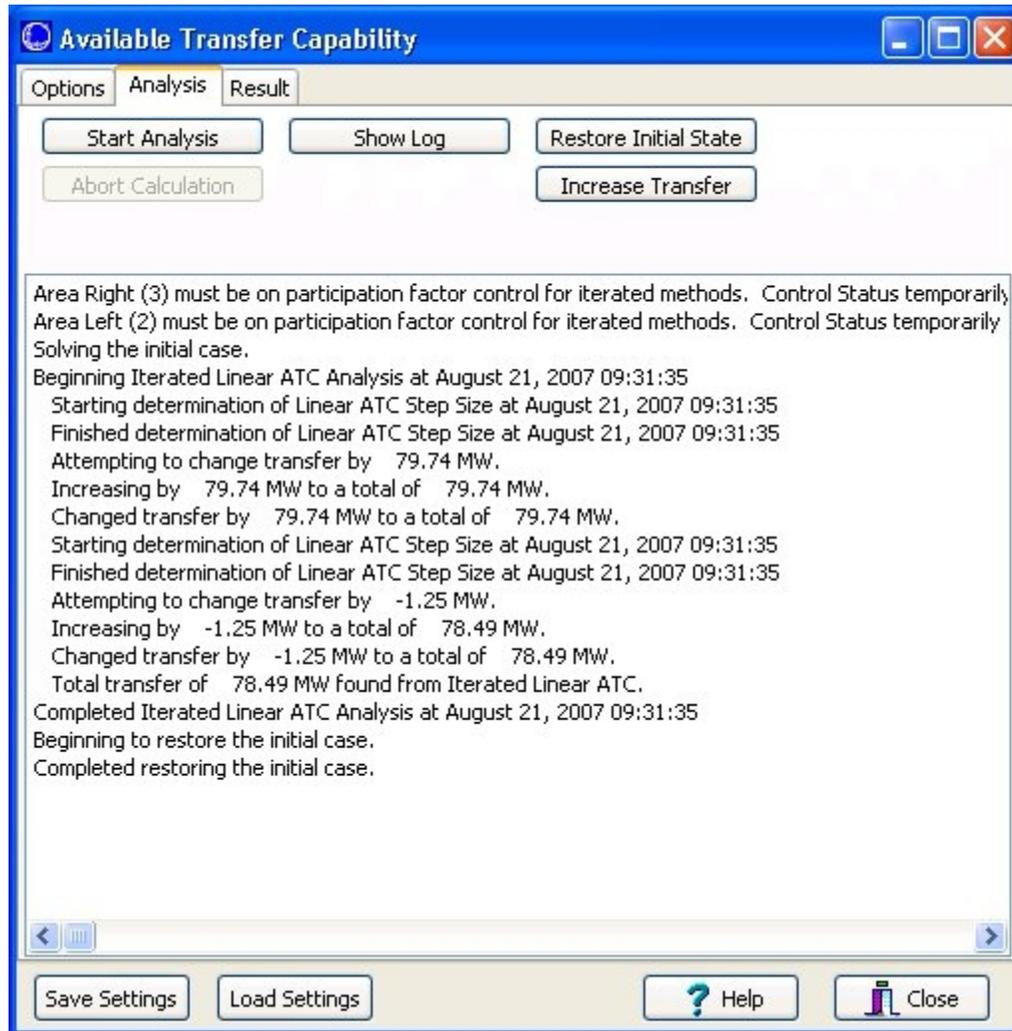
At the bottom of the dialog are buttons for 'Save Settings', 'Load Settings', 'Help', and 'Close'.

The Transfer Limiters Display contains tabbed sheets containing information on Branch, [Interface](#), and [Nomogram](#) Limiters. The user can also choose to display All Limiters. See [Transfer Limiters](#) for more information.

The Contingency Definition section displays information on the limiting contingency for selected transfer limiter if the user checked **Include Contingencies** on the [Common Options](#) tab.

ATC Dialog: Analysis

The Analysis tab, found on the [Available Transfer Capability dialog](#), is used to control the analysis process. The scrollable window displays a record of user-initiated actions relating to the analysis.



The Analysis tab has the following controls:

Start Analysis

The **Start Analysis** button begins the ATC Analysis using the settings and options defined by the user in the ATC Analysis Dialog. Progress of the ATC Analysis is shown in the scrollable pane of the ATC Dialog – Analysis Tab window.

Abort Calculation

When using one of the iterative ATC Solution Methods, or while analyzing Multiple Scenarios, click this button to abort the calculation.

Note: This does not immediately abort the solution. Simulator must restore the system state before completing the abort.

Show Log

Click this button to show the [Power Flow Solution Message Log](#).

Restore Initial State

Click this button to restore the system state to the state when the dialog was first opened.

Increase Transfer

Click this button to open the Ramp Transfer Up dialog in order to increase (or decrease) the transfer level manually. This transfer remains in the system state if the dialog is closed without restoring the initial state.

Additional analysis can be done on a case that has had the transfer increased by using the ATC tool; however, the control method used in the ATC tool may differ from that in use once the ATC tool has been exited. When injection groups are used to increase the transfer, the AGC method that is used is island-based AGC. The AGC method is then restored to what it was prior to increasing the transfer. If this same method is not used for subsequent analysis, the dispatch and system state will be modified from the state with the transfer applied. When using areas or super areas for increasing the transfer, the area or super area is put on participation factor control for the ramping.

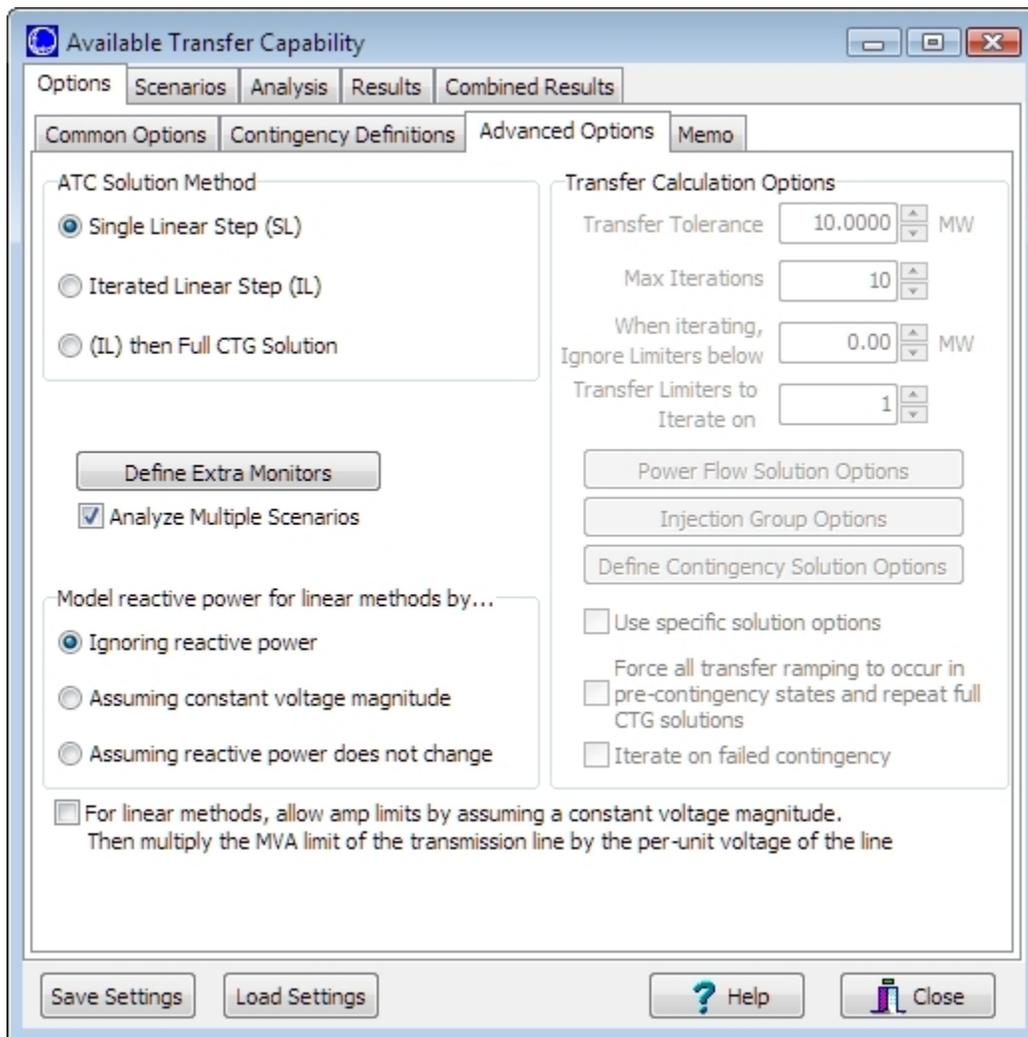
Areas/super areas participating in the transfer will remain on participation factor control following the ramping, but the same problem can exist as with the injection groups if the same AGC method is not in use with subsequent analysis after the transfer ramping.

If using areas or super areas for increasing the transfer, a transaction is added to the [MW Transactions](#) to implement the transfer. This transaction can be identified by the special ID that is assigned to it, *TEMP_ATC_TRANSACTION*. This transaction will remain in the case if closing the ATC dialog without restoring the initial state.

Multiple Scenario Available Transfer Capability Dialog

To perform [Available Transfer Capability Analysis](#) for several system scenarios, check **Analyze Multiple Scenarios** on the **Advanced Options** tab of the [Available Transfer Capability Dialog](#). When Analyze Multiple Scenarios is not checked, the available tabs are [Options](#), [Analysis](#) and [Result](#). When Analyze Multiple Scenarios is checked then the following changes occur:

- [Scenarios](#) tab appears
- [Results](#) tab appears
- Result Tab is removed (replaced by Results)
- [Combined Results](#) tab appears



By defining multiple scenarios, Simulator allows you to calculate ATC values for several different power system states automatically. Scenarios can be modified along three axes:

- Line Rating/Zone Load Scenarios (weather-related scenarios)

- Generation Scenarios (generation profiles)
- Interface constraints

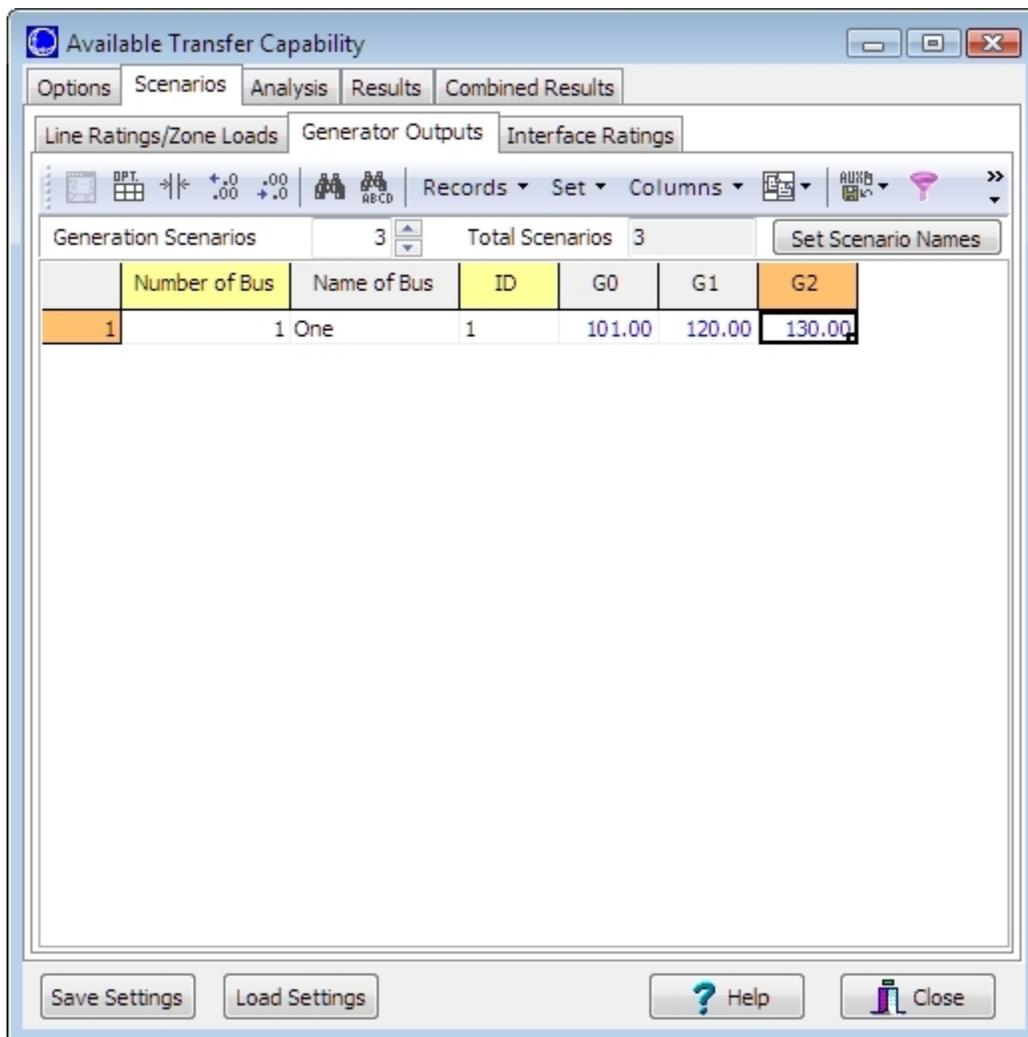
See [Scenarios Tab](#) for more information.

Multiple Scenario ATC Dialog: Scenarios

The Scenarios tab of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability Dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability Dialog](#).

By defining multiple scenarios, Simulator allows you to calculate ATC values for several different power system states automatically. Scenarios can be modified along three axes:

- Line Rating/Zone Load Scenarios (weather-related scenarios)
- Generation Scenarios (generation profiles)
- Interface constraints



Each tab contains a list of the power system elements that will be modified during different scenarios. These lists are a familiar [Case Information Display](#) providing the same functionality as other displays.

To insert a new power system element into the list, right-click on the list (below the headings) and choose **Insert**.

The Scenarios Tab has the following controls:

Tabs and Sub-Tabs

The Scenarios Tab contains three tabbed sheets: **Line Ratings/Zone Loads**, **Generator Outputs** and **Interface Ratings**.

Line Ratings and Zone Loads are varied together when analyzing scenarios. This was chosen because they often vary together as a function of the weather. Notice that three sub-tabs (labeled Line Ratings A, Line Ratings B, and Zone Loads) appear at the bottom of the Line Ratings/Zone Loads tab. The A rating is typically used as a base case ratings, whereas the B limit is typically used as a line rating under contingency. The way line ratings are utilized is defined in [Limit Monitoring Settings](#).

Number of Defined Scenarios per Element Type

On each tab, you may enter how many different scenarios should be defined for that kind of power system element. For instance if you set Generation Scenarios to 5, then list display on the Generation Tab will provide 5 columns labeled G0, G1, G2, G3, and G4. Generation outputs should then be entered into each cell representing the generation output in each scenario.

Total Scenarios

Once you have specified the scenarios, Simulator is able to perform ATC Analysis on every combination of the axes. For example, assume you have the following:

- 10 sets of line ratings and zones load
- 8 sets of generation profiles
- 3 interface constraints

This yields a total of 240 different scenarios to calculate ($10 \times 8 \times 3 = 240$). Be warned that that the more scenarios you analyze the longer the computation will take.

Set Scenario Names

This button will bring up the Scenario Names dialog, where the user is able to assign a different name to the scenarios.

Assume Constant Power Factor

This option is only available in the **Zone Loads** sub-tab of the **Line Ratings/Zone Loads** tab. If selected, it specifies that the Mvar will be ramped as the load MW is ramped, maintaining a constant power factor.

No Change in MVAR

This option is only available in the **Zone Loads** sub-tab of the **Line Ratings/Zone Loads** tab. If selected, it specifies that the load Mvar is kept constant, independently of the variations of the load MW.

Modified Elements

Each tab contains a list of the power system elements that will be modified during different scenarios. These lists are a familiar [Case Information Display](#) providing the same functionality as other displays.

To insert a new power system element into the list, right click on the list (below the headings) and choose **Insert**.

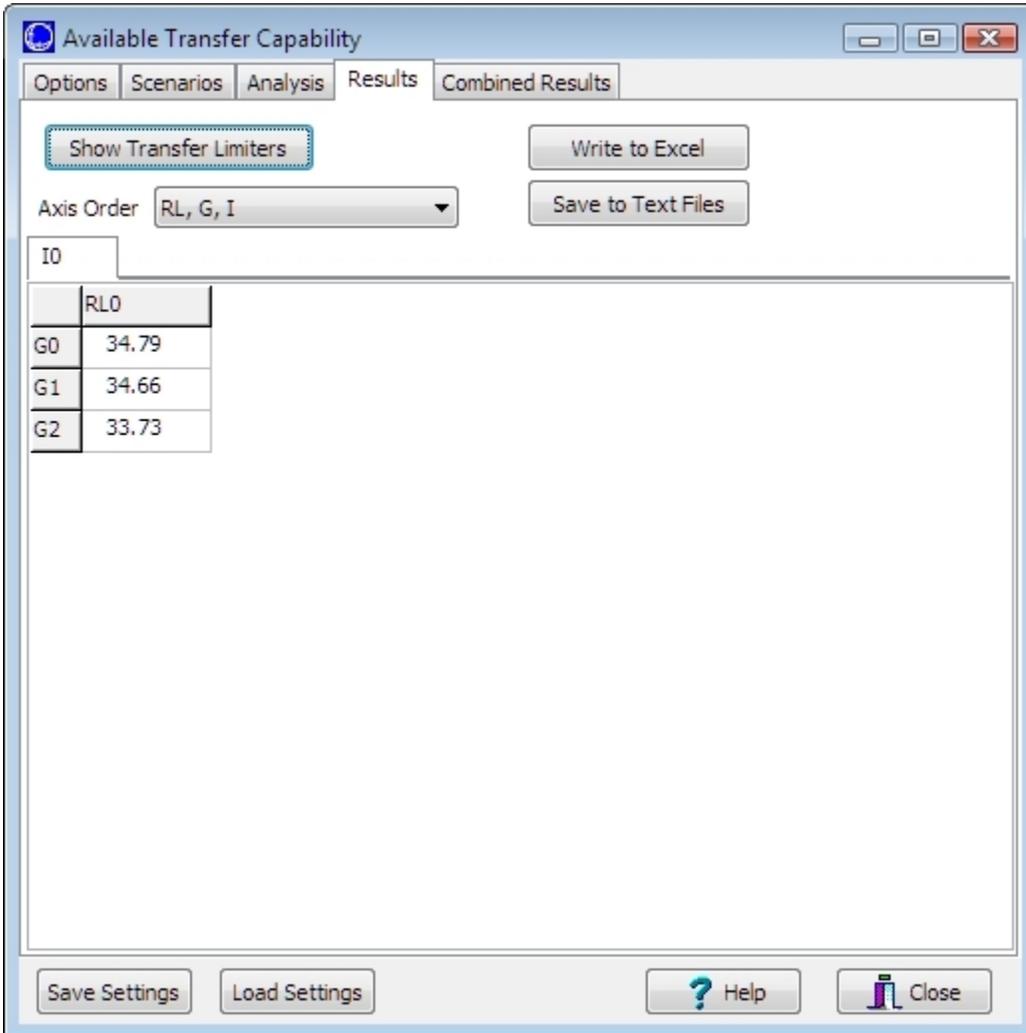
If a power system element does not show up in one of the lists of elements for a particular scenario, then it will retain its base case value during that scenario.

When saving or loading scenario elements in an auxiliary file, the elements are usually identified by primary key fields. Optionally, these elements can be identified by label. See the [Label](#) topic for more information about this.

Multiple Scenario ATC Dialog: Results

The Results tab of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability Dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability Dialog](#).

Once you have defined your scenarios and started the ATC Analysis, you can switch to the Results Tab to see the progress that ATC Analysis is making.



The primary part of the Results tab contains a spreadsheet-like look-up table display. The layout of the display is dictated by the Axis Order selected. Note that because there are three axes possible, the row and column entries represent the first two axes while the third axis is represented by tabs at the bottom of the table. See [Local Menu Options](#) for information regarding the options available when right-clicking within the table.

This dialog has the following controls:

Show Transfer Limiters Button

Click the **Show Transfer Limiters** button to view the [Transfer Limiters](#) found under each scenario. This will open a separate dialog that displays a list of the Transfer Limiters. To see the Transfer Limiters for a particular scenario, click on the workbook cell that represents the scenario you are interested in and the separate dialog will update appropriately.

Axis Order

This menu is used to select the desired axis order. The three axes correspond to the three [Scenarios Tabs](#).

- One axis has heading labels G0, G1, ... for the Generation Scenarios,
- Another with heading labels RL0, RL1, ... for the Rating/Load Scenarios,
- A third with heading labels I0, I1, ... for the Interface Scenarios.

Results Display

The primary part of the Results tab contains a spreadsheet-like look-up table display. The values shown in the table for each scenario are the most limiting Transfer Limit in MW.

Write to Excel

This button will send the results to an Excel spreadsheet.

Save to Text Files

This option will allow the user to save the results in an auxiliary file.

Multiple Scenario ATC Analysis Results: Local Menu Options

The [Results](#) tab of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability Dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability Dialog](#).

Right-clicking on the table contained on the Results tab will bring up a local menu containing several other options. The first several options are only enabled if a cell that represents a scenario is right-clicked. These options will perform an operation with respect to the Scenario:

Take me to Scenario ...

Implements the changes in the Scenario resulting in a modified system state.

Determine Transfer Limit For Scenario ...

Calculates the ATC for the Scenario, and then sets the system state back to the Initial State.

Take me to the Transfer Limit For Scenario ...

Performs ATC analysis for the Scenario, and then ramps the transfer to this limit.

Other options on this local menu are not related to the Scenario that has been clicked.

Increase Transfer for Present System State

Increments the transfer level for the present system state by a user-defined amount.

Return to Initial State

Returns the system state to the Initial State.

Multiple Scenario ATC Dialog: Combined Results

The Combined Results tab of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability Dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability Dialog](#).

The screenshot shows the 'Available Transfer Capability' dialog box with the 'Combined Results' tab selected. The main table displays transfer limitations for various branches and interfaces across different scenarios. Below this, a 'Contingency Definition' section shows a table with one entry: '1 OPEN Line Six 138.0 (6) TO Seven 138.0 (7) CKT 1' with a status of 'CHECK'. The dialog also includes 'Save Settings', 'Load Settings', 'Help', and 'Close' buttons.

	ATCScenario Line/Zone Scenario Number	ATCScenario Gen Scenario Number	ATCScenario Interface Scenario Number	Trans Lim	Limiting Element	Limiting CTG
17	0	0	0	400.00	Branch Six (6) TO Sev_L_000002Two-000006Six	
18	0	0	0	400.23	Branch Six (6) TO Sev_L_000007Seven-000005F	
19	0	0	0	400.23	Branch Six (6) TO Sev_L_000007Seven-000005F	
20	0	0	0	401.17	Branch Six (6) TO Sev_L_000002Two-000005Fiv	
21	0	1	0	34.66	Branch Two (2) TO Fir_L_000006Six-000007Sev	
22	0	1	0	34.66	Branch Two (2) TO Fir_L_000006Six-000007Sev	
23	0	1	0	37.14	Branch Two (2) TO Fir_L_000004Four-000005Fiv	
24	0	1	0	52.12	Branch Two (2) TO Fir_L_000002Two-000004Fou	
25	0	1	0	55.39	Branch Two (2) TO Fir_L_000002Two-000003Thr	
26	0	1	0	60.18	Branch Four (4) TO Fir_L_000002Two-000005Fiv	
27	0	1	0	178.89	Branch One (1) TO Tr_L_000002Two-000003Thr	

	Actions	Model Criteria	Stat
1	OPEN Line Six 138.0 (6) TO Seven 138.0 (7) CKT 1		CHECK

The tables on the Combined Results tab show the transfer limitations under ALL scenarios. The scenarios are identified by the default fields ATCLineZoneChanges, ATCGenChanges, and ATCInterfaceChanges. When defining [scenarios](#) for Line Ratings/Zone Loads, Generator Outputs, and Interface Ratings each corresponding scenario is given an integer identifier. The entries in the table fields correspond to these integer identifiers. Transfer limitations can be filtered based on the type of limitation by switching between the All Limiters, Branch Limiters, Interface Limiters, and Nomogram Interface Limiters sub-tabs, but the tables on any of these sub-tabs will still contain the relevant type of limiters from all scenarios. The transfer limiter results and remaining fields correspond to those described in the [Transfer Limiters Display](#) section.

The ATCLineZoneChanges, ATCGenChanges, and ATCInterfaceChanges fields can be used when interacting with the ATC results using auxiliary files and SimAuto to get access to the multiple scenario

results. These fields should be used with the TRANSFERLIMITER data type to specify the ATC scenario to which a transfer limitation corresponds. These fields are extra key fields for this data type. If these fields are blank or omitted from input data, the limitations are assumed to go with the standard single set of ATC results.

ATC Dialog: Transfer Limiters Display

The Transfer Limiters Display shows the results of [ATC Analysis](#). This display appears either on the [Result Tab](#) of the [Available Transfer Capability Dialog](#) or as a separate window when performing [Multiple Scenario Available Transfer Capability Analysis](#).

The display contains four different tabs: All Limiters, Branch Limiters, Interface Limiters, and Nomogram Interface Limiters.

The Contingency Definition Display at the bottom of the window shows information on the defined contingency that caused the limitation selected from one of the four tabbed pages of limiters. For more information on this display, see [Contingency Analysis - Contingency Definition Display](#).

The screenshot shows the 'Available Transfer Capability' dialog box. The 'Result' tab is selected, displaying a table of transfer limiters. The table has columns for 'Trans Lim', 'Limiting Element', 'Limiting CTG', '% OTDF', 'Pre-Trans Est', and 'Limit Used'. Below the table is a 'Contingency Definition' section with a table of actions.

	Trans Lim	Limiting Element	Limiting CTG	% OTDF	Pre-Trans Est	Limit Used
1	34.46	Branch Two (2) TO Five (5)	CL_00006Six-00007SevenC1	39.01	86.56	100.00
2	34.46	Branch Two (2) TO Five (5)	CL_00006Six-00007SevenC2	39.01	86.56	100.00
3	35.05	Branch Two (2) TO Five (5)	CL_00004Four-00005FiveC1	33.33	88.32	100.00
4	53.19	Branch Two (2) TO Five (5)	CL_00002Two-00004FourC1	27.57	85.34	100.00
5	55.31	Branch Four (4) TO Five (5)	CL_00002Two-00005FiveC1	21.45	48.14	60.00
6	55.77	Branch Two (2) TO Five (5)	CL_00002Two-00003ThreeC1	27.22	84.82	100.00
7	291.46	Branch Four (4) TO Five (5)	CL_00006Six-00007SevenC1	14.65	17.30	60.00
8	291.46	Branch Four (4) TO Five (5)	CL_00006Six-00007SevenC2	14.65	17.30	60.00
9	307.51	Branch One (1) TO Three (3)	L_00002Two-00003ThreeC1	3.23	55.05	65.00
10	337.03	Branch One (1) TO Three (3)	L_00002Two-00005FiveC1	4.49	49.87	65.00
11	353.02	Branch Four (4) TO Five (5)	CL_00001One-00002TwoC1	9.59	26.13	60.00

	Actions	Model Criteria	Status	Commen
1	OPEN Branch Six_138.0 (6) TO Seven_138.0 (7) CKT 1		CHECK	

Transfer Limiters Display

The Transfer Limiters Display has the following controls:

All Limiters Tab

The All Limiters tab shows a list of all the transfer limitations found. This includes limitations on branches, interfaces, areas, zones, etc...

Branch Limiters Tab

The Branch Limiters tab only shows those limitations with a transmission line or transformer as the limiting element. The limiting element description for the Branch Limiters tab is replaced by the bus numbers and names of the limiting element.

Interface Limiters Tab

The Interface Limiters Tab only shows those limitations with an interface as the limiting element. The limiting element description is replaced by the name of the interface.

Nomogram Interface Limiters Tab

The Nomogram Interface Limiters Tab only shows those limitations with a nomogram interface as the limiting element. The limiting element description is replaced by the name and segment of the nomogram interface.

Limiters Fields

Transfer Limit (shown by default)

The Transfer Limit Field shows the Transfer Limit in MW for the Limiting Element during the Limiting Contingency. This value depends on the ATC Solution Method Used:

- **Single Linear Step (SL):** Only one Linear ATC step is performed. The Transfer Limit values are those found during this step.
- **Iterated Linear Step (IL):** The Single Linear Step method is iterated during this method. The Transfer Limiters shown are those found during the final step performed. The actual Transfer Limit values are the values found at the last step plus the accumulated amount the transfer has been ramped. The Transfer Limitation(s) that were iterated on individually are highlighted in yellow.
- **Iterated Linear Step (IL) then Full CTG:** The Transfer Limits shown for the Transfer Limitation(s) that were used when iterating are equal to the accumulated amount that the transfer was ramped during the initial Iterated Linear Step and then the contingency analysis step. These limitations are highlighted in cyan. The Transfer Limits shown for all other limiters are the results of the initial Iterated Linear Step calculation performed before iterating on individual limitations for the contingency analysis.

For the iterated methods there are color indicators used to highlight the Transfer Limitations when certain situations are encountered:

- Fuchsia – Indicates that the power flow failed to converge as the transfer is ramped out in the iterated methods. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. When this situation is encountered, the top five dV/dQ sensitivities of all monitored buses in the case are also reported in the Transfer Limiters list. Limiting Element field will contain *POWERFLOW DIVERGENCE*.
- Lime Green – Indicates that the number of iterations has exceeded the maximum number of allowed iterations. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. Limiting Element field will contain *OSCILLATING*.
- Yellow – Indicates the Transfer Limitation(s) that were iterated on individually during the Iterated Linear Step (IL) method. Iteratively Found field will contain *YES*.
- Cyan – Indicates the Transfer Limitation(s) that were iterated on individually during the Iterated Linear Step (IL) then Full CTG method. Iteratively Found field will contain *FULL*.
- Orange – Indicates that the contingency failed to solve at some point during the Iterated Linear Step (IL) then Full CTG method. The Transfer Limit value that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. Iteratively Found field will contain *CTG_FAIL_IN_FULL*.
- Gray – Indicates that the power flow failed to converge while ramping the transfer for individual limiters. This type of limiter will not be reported if the power flow fails to converge when iterating on all limiters. The Transfer Limit that is reported is the accumulated transfer

amount that has been ramped as of the last successful solution. When this situation is encountered, the top five dV/dQ sensitivities of all monitored buses in the case are also reported in the Transfer Limiters list. The Iteratively Found field will further indicate the type of ramping limitation. If this limitation occurs when using the Iterated Linear Step (IL) method, the Iteratively Found field will be set to *RAMP_FAIL_IN_BASE*. The Iteratively Found field will also be set to *RAMP_FAIL_IN_BASE* if using the Iterated Linear (IL) then Full CTG method, the limiter that is being iterated on is a base case limitation with no contingency, and the power flow fails to converge during the ramping. If the power flow fails to converge during the ramping when using the Iterated Linear Step (IL) then Full CTG method during the Iterated Linear Step (IL) then Full CTG method while forcing the transfer ramping to occur pre-contingency, the Iteratively Found field will be set to *RAMP_FAIL_IN_FULL*. This indicates that the solution failed because of the transfer amount being ramped and not because the contingency failed to solve.

- Red - Indicates that the contingency failed to solve at zero transfer level (base case failure) when using the Iterated Linear (IL) then Full CTG method and choosing to iterate on failed contingencies. Iteratively Found field will contain *CTG_FAIL_IN_BASE*. The Transfer Limit value that is reported is the accumulated transfer amount that was ramped during the iterated process on the individual limiter without the contingency implemented.

Limiting Element (shown by default)

Shows a text description of the limiting element.

Limiting CTG (shown by default)

Shows the name of the limiting contingency.

% OTDF (shown by default)

This is the OTDF (or PTDF if the Limiting CTG is *Base Case*) on the Limiting Element for the transfer direction that is being studied. In other words, this is a linear estimate of the percent of the transfer that will appear on the Limiting Element if the Limiting CTG occurs.

Note: For iterated techniques, this is the PTDF or OTDF at the last Linear Iteration.

Pre-Transfer Value Estimate (shown by default)

If a contingency is not included in the Limiter, this is equal to the Initial Value. When a contingency is included in the Limiter, this is the linear estimate of the post-contingency flow before any transfer occurs. See [Available Transfer Capability Analysis](#).

Note: For iterated techniques, this is the estimate at the last Linear Iteration.

Limit Used (shown by default)

This is the value of the Limit being used by the ATC for the Limiting Element during the Limiting CTG. It reflects what is specified in [Limit Monitoring Settings](#) and options set on the [Advanced Options](#) tab.

MVA the Limit Used is based on

This is the unaltered MVA limit that is defined with a particular limiting element. This is the starting point for determining the **Limit Used** that reflects [Limiting Monitoring Settings](#) and options that are set on the [Advanced Options](#) tab.

Iteratively Found

String indicating if the limitation was found using one of the [iterated methods](#). Possibilities for this are *NO*, *YES*, *FULL*, *CTG_FAIL_IN_FULL*, *CTG_FAIL_IN_BASE*, *RAMP_FAIL_IN_BASE*, and *RAMP_FAIL_IN_FULL*. The various entries correspond to the limit being highlighted in different colors. Additional information about what the different entries mean is given above in the **Transfer Limit** description.

Contingency Definition Display

This display shows information on the defined contingency that caused the limitation selected from one of the four tabbed pages of limiters. For more information on this display, see [Contingency Analysis - Contingency Definition Display](#).

ATC Analysis Methods: Solution Methods

Simulator provides three methods of determining ATC.

- [Single Linear Step \(SL\)](#)
- [Iterated Linear Step \(IL\)](#)
- [Iterated Linear Step \(IL\) then Full Contingency Solution](#)

ATC Analysis Methods: Single Linear Step (SL)

The Single Linear Step approach is the most common ATC method. This method of ATC analysis uses only information about the present system state and sensitivities (mathematical derivatives) about the present system state. These sensitivities are embodied in the [PTDF](#) and [LODF](#) calculations.

Consider a transmission line with a limit of 10 MW, present loading of 5 MW and a PTDF of 10%. The estimated maximum transfer without causing an overload on the line is

$$\text{Transfer Limitation} = (\text{Limit} - \text{Present Loading}) / \text{PTDF} = (10 - 5) / 0.1 = 50 \text{ MW}$$

When including contingency analysis, the [OTDF \(Outage Transfer Distribution Factor\)](#) and linearized estimates of post-contingency flows are used to determine the Transfer Limitation.

$$\text{Transfer Limitation} = (\text{Limit} - \text{Post-Contingency Loading}) / \text{OTDF}$$

If we find the Transfer Limitation for every transmission branch (and interface) during each contingency, then the ATC is equal to the smallest Transfer Limit value.

Contingency elements have an associated [Status](#) that determines under what conditions a particular action will be implemented. When ATC values are calculated using the single linear step method, contingency actions are not actually implemented. Because contingency actions are not actually implemented, *POSTCHECK* actions cannot be applied after other actions have been performed and the power flow has been solved. Instead, *POSTCHECK* actions act as *CHECK* actions during the single linear step method. Whether or not model criteria are met for *CHECK* and *POSTCHECK* contingency actions will be determined at the zero transfer level rather than at the transfer level that determines the limiting transfer.

Note: Simulator also monitors the possibility that a transfer will reduce the flow on a line until the line reaches its limit for flow in the opposite direction.

ATC Analysis Methods: Iterated Linear Step (IL)

The Single Linear Step is an extremely fast method for determining the ATC. However, because it only uses present operating point information, controller changes are not taken into account. The linearization assumes that all controllers are fixed. The Iterated Linear Step (IL) method provides an alternative to the Single Linear Step that allows controller changes, but still performs its analysis in a reasonable amount of time. The (IL) method operates as follows:

- Perform Single Linear Step
- Set the Stepsize = Minimum Transfer Limitation found in Single Linear Step which is greater than the specified **When iterating, Ignore Limiters below** value
- If [$\text{abs}(\text{Stepsize}) \leq \text{Tolerance}$] or [$\text{Iterations} \geq \text{Max Iterations}$] then stop
 - If the maximum number of iterations is reached, a transfer limitation will be reported highlighted in lime green with the Iteratively Found field set to *OSCILLATING* indicating that the iterated process is oscillating. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
 - The iteration count is reset at the beginning of the iteration process.
- Implement transfer by amount of Stepsize and resolve power flow
 - The power flow is solved according to the options set with the [Power Flow Solution Options](#).
 - When calculating a transfer between injection groups, the [Island-Based AGC Power Flow Solution Options](#) applicable to dispatch using an injection group are used when determining which generators and loads will participate in the transfer. These options include whether or not AGC status should be considered when determining which units will participate in the ramping, whether or not to enforce generator MW limits, and whether or not to allow negative loads. The power factor to use when adjusting load is also specified with these options.
 - To prevent oscillation of the solution, the Stepsize may be limited if the sign of the Stepsize changes throughout the iterations. At each iteration, the direction of the Stepsize is checked and the Stepsize is bounded based on the accumulated transfer at each iteration. The accumulated transfer is the total amount that the transfer between the seller and buyer has been ramped. The accumulated transfer is updated with the current Stepsize once it has been determined that Stepsize can be achieved. At the start of the iterated process, LowBound is initialized to a large negative number and HighBound is initialized to a large positive number. At each iteration the following checks are performed:
 - If $\text{Stepsize} > 0$ then $\text{LowBound} = (\text{Accumulated Transfer})$
 - Else If $\text{Stepsize} < 0$ then $\text{HighBound} = (\text{Accumulated Transfer})$
 - If $\text{Stepsize} \geq (\text{HighBound} - \text{Accumulated Transfer}) * 0.7$
 - Then $\text{Stepsize} = (\text{HighBound} - \text{Accumulated Transfer}) * 0.7$
 - Else If $\text{Stepsize} \leq (\text{LowBound} - \text{Accumulated Transfer}) * 0.7$
 - Then $\text{Stepsize} = (\text{LowBound} - \text{Accumulated Transfer}) * 0.7$
 - During the implementation of the transfer, if the full Stepsize cannot be achieved, an attempt is made to implement as much of the transfer as possible in smaller stepsizes. The first attempt is to cut the Stepsize in half. If that succeeds then an iterated process occurs where the difference between the original full Stepsize and the successful stepsize is ramped. This process continues with the incremental stepsizes until as much of the original Stepsize ramping as possible is achieved.
 - If the power flow fails to converge at this step, a transfer limitation will be reported highlighted in fuchsia with the Iteratively Found field set to *POWERFLOW DIVERGENCE*. The Transfer Limit that is reported is the accumulated transfer amount that has been

ramped as of the last successful solution. This transfer limitation means that the full desired transfer amount cannot be achieved.

- o If the full transfer amount cannot be achieved in this process when all limiters are being iterated on, the iterative process will continue and iterate on individual limiters.
- o If the full transfer amount cannot be achieved in this process when individual limiters are being iterated on, the analysis method selected will dictate how the process continues. If using the Iterated Linear Step (IL) method, a Transfer Limit will be reported that is the accumulated transfer amount that has been ramped as of the last successful solution and this limiter will be highlighted in gray with the Iteratively Found field set to *RAMP_FAIL_IN_BASE*. If the Iterated Linear Step (IL) method is being used as part of the Full CTG Solution method, the process will continue into the contingency solution step unless the limiter that is being iterated on is a base case limitation. If the limiter is a base case limitation, a Transfer Limit will be reported the same as if using the Iterated Linear Step (IL) method.
- o The ramping of the transfer is done so that each step adjusts injection from a common starting point so that the ramping is effectively done in the same direction even if the Stepsize backs off the transfer. At the start of the iterated process, the base case generation and load values are stored. At each step where transfer ramping occurs, the generation and load values are returned to their base case levels and the transfer ramping is implemented from that point. As an example, take the situation where 1000 MW is ramped. The next Stepsize is calculated to be -100 MW. The total transfer amount achieved is then 900 MW. If ramping does not proceed in the same direction, generators that hit maximum limits when ramping out to 1000 MW will back off from these limits when ramping the -100 MW Stepsize. Ramping out to the total transfer amount of 900 MW in one step causes generators to hit their limits and stay there. If the ramping does not proceed in the same direction, ramping out by the total amount in one step and ramping by backing off the transfer will yield different results because some generators will no longer be at limits when ramping in steps. Ramping in the same direction is intended to prevent this discrepancy with generators at limits.
- o Reserve limits on generators and loads participating in the transfer will be checked if the option to enforce limits is active for the selected transactors. Reverse limits are checked regardless of how the option to **Report Generation Reserve Limits** is set with the [Common Options](#). For areas and super areas this means that the global enforcement of generator limits is active, the area enforcement of limits is active, and there is at least one generator whose enforcement of limits is active and this generator is on AGC control. For injection groups this means that the enforcement of limits for the injection group is active. When a reserve limit is found, the current iterations will stop and a transfer limit will be reported with the reserve limit and the area, super area, or injection group that is at a limit. If a reserve limit is encountered during the step where all transfer limiters are iterated on together, the entire ATC process will stop. If a reserve limit is encountered during the step where a limiter is being iterated on individually, the iterations will only stop for that limiter; other limiters will be processed if they do not hit a reserve limit as well.
- At new operating point, go back to first step and repeat

As the initial iteration process is performed, the transfer limiter may be different (i.e. the contingency and limiting element pair may be different) during each iteration. Once the Stepsize is determined to be less than or equal to the tolerance (or the maximum number of iterations is met) the order of the limiters is set.

At this point, the first user specified **Transfer Limiters to Iterate On** are taken from the list of limiters and the above process is repeated for each limiter individually, with the contingency and limiting element pair of each limiter remaining set as determined in the initial process. Some limiters that will appear in the list of limiters are not valid for iterating on individually. These limiters include those indicating the full transfer amount could not be ramped in the initial process and those indicating

that a reserve limitation has been met on the source or sink. These type of limiters will be skipped when choosing those to iterate on individually. Transfer limitations for transfer limits found by iterating on them individually will be highlighted in yellow in the results and the Iteratively Found field will be set to *YES*.

This method takes into account controller changes that occur as you ramp out to the transfer level, but still avoids the full simulation of contingencies.

This method can be applied between combinations of areas and super areas, OR between two injection groups. Combinations of areas/superareas and injection groups are not allowed.

ATC Analysis Methods: Iterated Linear Step (IL) then Full CTG Solution

This method takes the solution process a step further than just the Iterated Linear (IL) method. Instead of using linear sensitivities to calculate the impact of contingencies on the transfer limits, full contingencies are implemented and solved with a full power flow solution. Be aware however, that this calculation method can be extremely slow.

The initial iteration process of (IL) then Full CTG Solution method starts by performing the steps outlined in the [Iterated Linear \(IL\)](#) process. As the initial iteration process is performed, the transfer limiter may be different (i.e. the contingency and limiting element pair may be different) during each iteration. Once the Stepsize is determined to be less than or equal to the tolerance (or the maximum number of iterations is met) the order of the limiters is set.

At this point, the first user specified **Transfer Limiters to Iterate On** are taken from the list of limiters and the steps outlined in the [Iterated Linear \(IL\)](#) process are repeated for each limiter individually, with the contingency and limiting element pair of each limiter remaining set as determined in the initial process. Some limiters that will appear in the list of limiters are not valid for iterating on individually. These limiters include those indicating the full transfer amount could not be ramped in the initial process and those indicating that a reserve limitation has been met on the source or sink. These type of limiters will be skipped when choosing those to iterate on individually.

Once the iterated process has been repeated for an individual limiter, there are two different ways that the processing of the full contingency can proceed. If choosing to **NOT Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions**, the contingency of each transfer limiter is then enforced in the power flow topology, and a power flow solution is performed to give a new contingency power flow solution to use as the base for each transfer limiter. If the option to **Use Specific Solution Options for Contingencies** is checked, then the power flow is solved with the options defined with the [Contingency Solution Options](#). Otherwise, the power flow is solved with the options specified with the [Power Flow Solution Options](#). In either case, the [Make-Up Power](#) option specified with the contingency analysis options is used when applying the contingency. The iterative process outlined in the [Iterated Linear \(IL\)](#) method is then repeated again for each individual transfer limiter with the specified contingency of each limiter actually in place in the power flow solution. If the contingency solution fails to converge, the transfer limitation will be reported highlighted in orange with the Iteratively Found field set to *CTG_FAIL_IN_FULL*, and the Transfer Limit that is reported will be the value determined from the iterated process performed on the individual limiter before the contingency was applied.

If choosing to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions**, after the iterated process is complete for an individual limiter, the contingency of each transfer limiter is fully implemented in the following steps:

- Store the system state following the iterated process on the individual limiter
 - At this point the accumulated transfer amount is the total amount that has been transferred following completion of the iterated process on the individual limiter.
 - The Stepsize is zero for the first iteration.
- Apply the contingency and perform full power flow solution
 - If the option to **Use Specific Solution Options for Contingencies** is checked, then the power flow is solved with the options defined with the [Contingency Solution Options](#). Otherwise, the power flow is solved with the options specified with the [Power Flow Solution Options](#). In either case, the [Make-Up Power](#) option specified with the contingency analysis options is used when applying the contingency.
 - If the option to **Iterate on failed contingency** is NOT checked and the power flow fails to solve, the iterated process for the individual limiter will stop and a transfer limitation will be reported highlighted in orange with the Iteratively Found field set to *CTG_FAIL_IN_FULL*.

The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. If the contingency fails to solve immediately following the completion of the iterated process on the individual limiter without the contingency implemented, the Transfer Limit that is reported is the accumulated transfer amount achieved from that process without the contingency implemented.

- o If the option to **Iterate on failed contingency** IS checked and the power flow fails to solve, a new iteration loop is entered to determine the highest transfer level at which the contingency can be solved. This loop attempts to determine a more accurate transfer level at which the contingency fails to solve rather than simply reporting the transfer level at which the last power flow was successful regardless of the current Stepsize. To start the iterative process, the current Stepsize is reduced by half. The ramping is performed and the contingency is solved. The Stepsize will remain the same as long as the contingency can be solved. If the contingency fails to solve, the Stepsize will be reduced by half again. The ramping will continue until the contingency no longer solves, the accumulated transfer during the process meets the original Stepsize, or the Stepsize becomes smaller than the Stepsize tolerance.
 - o An additional check is done if the contingency does solve during this process; the flow on the monitored element is checked to determine if it exceeds its limit. If it is overloaded, the Stepsize will be reduced by half until the monitored element is no longer over its limit. While this check is being done on the monitored element limit, the ramping continues out to the specified amount and the contingency is solved. During this entire process, the Stepsize is limited to be between zero and the Stepsize that was ramped when the contingency failed to solve for the first time. When the iterative process on the failed contingency completes, the iterations are done for this particular limiter. The Transfer Limit that is reported is the accumulated amount that is possible without overloading the monitored element and in which the contingency solves. The limiter will be reported highlighted in orange with the Iteratively Found field set to *CTG_FAIL_IN_FULL* if the monitored element is not loaded to its limit. If during this process, the monitored element is at its thermal limit and the contingency solves, the Iteratively Found field is set to *FULL* and the limiter will be highlighted in cyan.
 - o If the contingency fails to solve at any transfer level (base case failure), the limiter will be reported highlighted in red with the Iteratively Found field set to *CTG_FAIL_IN_BASE*. The Transfer Limit that is reported is the accumulated transfer amount that was achieved from the iterated process on the individual limiter without the contingency implemented.
 - o If the power flow solution is successful, update the accumulated transfer with the Stepsize.
- Perform Single Linear Step on the individual limiter
 - o The Stepsize is the Transfer Limit found for the individual limiter. If the Stepsize is less than the specified **When iterating, Ignore Limiters below** value minus the accumulated transfer amount, it is set to zero.
- If $[\text{abs}(\text{Stepsize}) \leq \text{Tolerance}]$ or $[\text{Iterations} \geq \text{Max Iterations}]$ then stop
 - o If the maximum number of iterations is reached, a transfer limitation will be reported highlighted in lime green with the Iteratively Found field set to *OSCILLATING* indicating that the iterated process is oscillating. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
 - o The iteration count is reset at the beginning of the contingency iteration process for each individual limiter.
- Restore system to the state stored in the first step
 - o This step removes the contingency that was implemented so that the transfer ramping occurs prior to the contingency being applied.
- Implement transfer by amount of Stepsize and resolve power flow
 - o The power flow is solved according to the options set with the [Power Flow Solution Options](#).

- o When calculating a transfer between injection groups, the [Island-Based AGC Power Flow Solution Options](#) applicable to dispatch using an injection group are used when determining which generators and loads will participate in the transfer. These options include whether or not AGC status should be considered when determining which units will participate in the ramping, whether or not to enforce generator MW limits, and whether or not to allow negative loads. The power factor to use when adjusting load is also specified with these options.
 - o To prevent oscillation of the solution, the Stepsize may be limited if the sign of the Stepsize changes throughout the iterations. At each iteration, the direction of the Stepsize is checked and the Stepsize is bounded based on the accumulated transfer at each iteration. The accumulated transfer is the total amount that the transfer between the seller and buyer has been ramped. The accumulated transfer is updated with the current Stepsize once it has been determined that Stepsize can be achieved. At the start of the iterated process, LowBound is initialized to a large negative number and HighBound is initialized to a large positive number. At each iteration the following checks are performed:
 - If Stepsize > 0 then LowBound = (Accumulated Transfer)
 - Else If Stepsize < 0 then HighBound = (Accumulated Transfer)
 - If Stepsize >= (HighBound - Accumulated Transfer)*0.7
 - Then Stepsize = (HighBound - Accumulated Transfer)*0.7
 - Else If Stepsize <= (LowBound - Accumulated Transfer)*0.7
 - Then Stepsize = (LowBound - Accumulated Transfer)*0.7
 - o During the implementation of the transfer, if the full Stepsize cannot be achieved, an attempt is made to implement as much of the transfer as possible in smaller stepsizes. The first attempt is to cut the Stepsize in half. If that succeeds than an iterated process occurs where the difference between the original full Stepsize and the successful stepsize is ramped. This process continues with the incremental stepsizes until as much of the original Stepsize ramping as possible is achieved.
 - o The ramping of the transfer is done so that each step adjusts injection from a common starting point so that the ramping is effectively done in the same direction even if the Stepsize backs off the transfer. Additional details about this are provided in the steps of the [Iterated Linear \(IL\)](#) process.
 - o If the power flow fails to solve at this point, the iterated process for the individual limiter will stop and a transfer limitation will be reported highlighted in gray with the Iteratively Found field set to *RAMP_FAIL_IN_FULL*. This Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
 - o Reserve limits on generators and loads participating in the transfer will be checked if the option to enforce limits is active for the selected transactors. Reserve limits are checked regardless of how the option to **Report Generation Reserve Limits** is set with the [Common Options](#). For areas and super areas this means that the global enforcement of generator limits is active, the area enforcement of limits is active, and there is at least one generator whose enforcement of limits is active and this generator is on AGC control. For injection groups this means that the enforcement of limits for the injection group is active. When a reserve limit is found, the current iterations will stop and a transfer limit will be reported with the reserve limit and the area, super area, or injection group that is at a limit.
- Go back to second step and repeat

Transfer limitations found by implementing the full contingency solution are highlighted in cyan in the results with the Iteratively Found field set to *FULL*.

This method can be applied between combinations of areas and superareas, OR between two injection groups. Combinations of areas/superareas and injection groups are not allowed.

When using this method, the PTDF and OTDF cutoff values are set to 0.01% internally by the ATC process when iterating on individual limiters. Once limiters are in the list as having acceptable distribution factors following the process where all limiters are iterated on as a whole, they should not be eliminated during the process when they are being iterated on individually. Actual implementation of the contingency as opposed to the linear determination of the impact of the contingency could lead to some large differences in the distribution factor post-contingency for some contingency scenarios. The only time that these differences could be a problem is if implementation of the contingency actually causes the monitored element to no longer be limiting, i.e. the distribution factor goes to zero. If this happens for any limiter, that limiter will be marked internally as having an infinite transfer amount, will show up in the results with a very large transfer limiter value, and will be excluded from the count of the number of limiters that have been iterated on. The next limiter in the list will then be processed until the number of **Transfer Limiters to Iterate on** has been met or there are no more limiters.

Simulator Automation Server (SimAuto) Overview

The SimAuto tool is only available if you have purchased the SimAuto add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the SimAuto version of Simulator.

PowerWorld Corporation also offers Optimal Power Flow ([OPF](#)), Available Transfer Capability [ATC](#), Simulation Automation Server ([SimAuto](#)), and Security Constrained Optimal Power Flow ([SCOPE](#)) add-ons. For more information see.

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on COM Objects and Automation Servers, see the help for Microsoft Windows.

The Automation Server of Simulator works very well in combination with Simulator [Script Commands](#) and [Auxiliary Files](#). It is beneficial to become familiar with these topics when considering using the Simulator Automation Server.

Previous users of SimAuto in Version 9 will need to update their function calls to SimAuto functions. PowerWorld Corporation found it imperative to change the function calls for SimAuto, in order to remedy irreconcilable problems when using SimAuto with some programming languages. The documentation provided should provide adequate help on the changes needed, but as always, if any questions arise, please [contact](#) PowerWorld Corporation for more information.

Note: When Simulator is launched using the SimAuto object, any currently running instances of Simulator launched manually will be unaffected. SimAuto will launch a separate background instance of Simulator so that any manually opened instances of Simulator can continue to be used by the user.

Installing Simulator Automation Server

Note that previous users of SimAuto in Version 9 will need to update their function calls to SimAuto functions. PowerWorld Corporation found it imperative to change the function calls for SimAuto, in order to remedy irreconcilable problems when using SimAuto with some programming languages. The documentation provided should provide adequate help on the changes needed, but as always, if any questions arise, please [contact](#) PowerWorld Corporation for more information.

Installing the [Simulator Automation Server](#) requires no additional steps beyond installing PowerWorld Simulator as normal. When a version of PowerWorld Simulator containing the Simulator Automation Server is installed on your computer, the install program automatically adds the information needed by the Simulator Automation Server to the registry.

If for some reason the registration fails, be sure you have the SimAuto add-on for Simulator and you have either run the application on the computer or run the command **pwrworld / regserver** at the command line. If instead you would like SimAuto to un-register itself as a COM object, run the command **pwrworld /unregserver** at the command line.

Including Simulator Automation Server Functions

Before you can access the [functions](#) defined by the [Simulator Automation Server](#) when writing the code for your external program, you must first include the library of functions defined for the Simulator Automation Server. This kind of library is referred to as a Type Library, which describes the available functions in a manner that can be interpreted by different programming languages. Importing a Type Library from another program is usually fairly simple, but the procedure does vary depending on the programming tool you are using. Please see the help for your programming tool of choice on how to import a Type Library or COM functions from another program.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In Delphi, choose **Import Type Library...** from the Project menu.
- In the list of libraries, search for and choose pwrworld Library.
- If pwrworld Library is not in the list, click **Add**. Find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click **Open**.
- You should see the class name TSimulatorAuto in the list of Class names.
- Click **Install** to include the PowerWorld Simulator Type Library.

Microsoft Visual Basic for Applications

- No additional tasks are necessary
- Importing Type Library still works (See [Including Functions for version 9](#)).

Microsoft Visual C++

- Install the version of PowerWorld Simulator with the Simulation Automation Server included.
- Add **#import "...\powerworld.exe"** in your external program code, using the full path to the PowerWorld Simulator executable program.
- Add **using namespace pwrworld** in your external program code.

Matlab v.6.5 r.13

- No additional tasks are necessary.

Connecting to Simulator Automation Server

Once the Type Library or COM functions have been included in your programming environment, the [Simulator Automation Server](#) can be handled as any other object in your code. The method for assigning and connecting to the Simulator Automation Server can vary depending on the programming environment used, but the idea is basically the same. You define a variable in your program to point to the server object, which is called SimulatorAuto. If the Type Library was imported properly, you should have full access to the SimulatorAuto object and its defined [functions](#). Again, the procedure for creating the object and connecting to SimulatorAuto may vary for different programming languages. Check the help for your programming environment on connecting to COM or Automation servers.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi 5

- Add **pwrworld_TLB** to the **uses** section of your unit.
- Declare a variable globally or as part of another object: **A : ISimulatorAuto**
- Initialize the variable: **A := nil**
- To connect to the Simulator Automation Server, create the connection: **A := CoSimulatorAuto.create**
- Perform function calls to the Simulator Automation Server: **Output := A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, remove the reference by again setting: **A := nil**

Microsoft Visual Basic for Applications

Early Binding:

- To connect to the Simulator Automation Server, create the connection initializing the variable:

Dim A as New pwrworld.SimulatorAuto

Late Binding:

- Declare a variable globally or as part of another object or function: **Dim A As Object**
- To connect to the Simulator Automation Server, create the connection:

Set A = CreateObject("pwrworld.SimulatorAuto")

Both Early and Late Binding:

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction parameters**
- To close the connection to the Simulator Automation Server, remove the reference: **Set A = Nothing**
- If Type Library was imported, connection can also be achieved as in version 9 (See [Connecting to Simulator Automation Server in version 9](#)).

Microsoft Visual C++

- Declare a variable globally or as part of another object or function: **IsimulatorAutoPtr *A**
- Declare a variable globally or as part of another object or function: **CLSID clsid**
- Declare a variable globally or as part of another object or function: **HRESULT hr**
- Obtain the class identifier (clsid) with the following command:

hr = CLSIDFromProgID(L"pwrworld.SimulatorAuto", &clsid)

- Initialize variable A: **A = new IsimulatorAutoPtr**
- To connect to the Simulator Automation Server, create the connection:

hr = A>CreateInstance(clsid, NULL, CLSCTX_SERVER)

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, release the reference:

hr = A>Release()

Matlab v.6.5 r.13

- To connect to the Simulator Automation Server, create the connection:

A = actxserver('pwrworld.SimulatorAuto')

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, delete the connection: **delete(A)**

Passing Data to the Simulator Automation Server

Passing Data to the Simulator Automation Server

All data to the Simulator Automation Server is passed by value rather than by reference (in pointer terminology, this corresponds to sending data instead of pointer to data; in Microsoft Visual Basic®, this corresponds to sending data ByVal instead of ByRef). This makes the marshalling of data between client software and the COM object much easier.

No Optional Parameters

There are no optional parameters for any of the Simulator Automation functions. All functions must be called with every argument filled.

Getting Data from the Simulator Automation Server

The Output Structure

Every function called on the SimulatorAuto object returns the same value, Output, which has a well-defined structure.

Output is of type **VARIANT**, and is an array of **VARIANT**s. Output is zero-indexed.

The first element *always* contains any errors occurring during execution. For those functions returning more than one element in the Output array (e.g. ListOfDevices), explanation is provided below when discussing the specific method.

Error Handling

As mentioned above, the first item in the Output VARIANT array, Output[0], contains any errors occurring during the function's execution. If no errors occurred during the function's execution, Output[0] will be set to an empty BSTR (string) represented in most languages by either "" or "".

Error Format

If an error string is returned, it will be in the following format:

[method name]: [error_explanation]

e.g. RunScriptCommand: Error occurred processing script command - check command syntax

Simulator Automation Server Functions

The following list of functions is currently available once the SimulatorAuto object is set in your code. Check the help sections on these functions to see more detail on the particular function.

[ChangeParameters](#)

[ChangeParametersSingleElement](#)

[ChangeParametersMultipleElement](#)

[ChangeParametersMultipleElementFlatInput](#)

[CloseCase](#)

[GetParametersSingleElement](#)

[GetParametersMultipleElement](#)

[GetParametersMultipleElementFlatOutput](#)

[GetParameters](#) (for compatibility with version 9)

[ListOfDevices](#)

[ListOfDevicesAsVariantStrings](#)

[ListOfDevicesFlatOutput](#)

[OpenCase](#)

[ProcessAuxFile](#)

[RunScriptCommand](#)

[GetFieldList](#)

[SaveState](#)

[LoadState](#)

[SaveCase](#)

[SendToExcel](#)

[WriteAuxFile](#)

ChangeParameters Function

The ChangeParameters function has been replaced by the [ChangeParametersSingleElement](#) function. ChangeParameters can still be called as before, but will now just automatically call ChangeParametersSingleElement, and pass on the parameters to that function.

Function Prototype

ChangeParameters(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified.

ChangeParametersSingleElement Function

The ChangeParametersSingleElement function allows you to set a list of parameters for a single object in a case loaded into the [Simulator Automation Server](#). In addition to changing parameters for objects, this function can also be used to set options for some of the Simulator tools, such as [ATC](#) and [OPE](#). This function is identical in setup to the [GetParametersSingleElement](#) function, with the exception that the Values array must contain a value for each field variable given in the ParamList array.

Function Prototype

ChangeParametersSingleElement(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified.

Output

ChangeParametersSingleElement only returns the first element in Output, the error string.

Notes

The ParameterList and Values arrays must be the same size, as each parameter must have a corresponding value to be assigned.

ChangeParametersSingleElement Sample Code

Borland® Delphi

```

Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the maximum number of iterations
// and the system base for the power flow simulations
ParamList := VarArrayCreate([1,2], varOleStr);
ParamList[1] := 'MaxItr';
ParamList[2] := 'SBase';

// ValueList is setup with 41 and 410 for MaxItr and SBase,
// respectively
ValueList := VarArrayCreate([1,2], varOleStr);
ValueList[1] := 41;
ValueList[2] := 410;

// Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement('Sim_Solution_Options', _
  ParamList, ValueList)

```

Microsoft® Visual Basic for Applications

```

' Set ParamList up to modify the maximum number of iterations
' and the system base for the power flow simulations
Dim ParamList As Variant
ParamList = Array("MaxItr", "Sbase")

' ValueList is setup with 41 and 410 for MaxItr and SBase,
' respectively
Dim ValueList As Variant
ValueList = Array(45, 90)

' Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement("Sim_Solution_Options", _
  ParamList, ValueList)

```

Matlab®

```

% Set ParamList up to modify the maximum number of iterations
% and the system base for the power flow simulations

```

```
ParamList = {'MaxItr' 'Sbase'};

% values is setup with 41 and 410 for MaxItr and SBase,
% respectively
values = [41 410];

% Convert the values matrix to a set of cells for passing
% through the COM interface
ValueList = num2cell(values);

' Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement('Sim_Solution_Options', _
ParamList, ValueList)
```

ChangeParametersMultipleElement Function

The ChangeParametersMultipleElement function allows you to set parameters for multiple objects of the same type in a case loaded into the [Simulator Automation Server](#). This function is very similar to the [ChangeParametersSingleElement](#), but allows for modifying multiple elements with a single function call. The advantage of this function is that it is much faster to change multiple elements with a single function call than it is to repeatedly call ChangeParametersSingleElement multiple times.

Function Prototype

ChangeParametersMultipleElement(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

ValueList : Variant A variant array storing arrays of variants. This is the difference between the multiple element and single element change parameter functions. This array stores a list of arrays of values matching the fields laid out in ParamList. You construct ValueList by creating an array of variants with the necessary parameters for each device, and then inserting each individual array of values into the ValueList array. SimAuto will pick out each array from ValueList, and calls ChangeParametersSingleElement internally for each array of values in ValueList.

Output

ChangeParametersMultipleElement only returns the first element in Output, the error string.

Notes

The ParameterList and each array of values stored in ValueList must be the same size, as each parameter must have a corresponding value to be assigned.

ChangeParametersMultipleElement Sample Code

Borland® Delphi

```

Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the MW output of generators
ParamList := VarArrayCreate([1,3], varOleStr);
ParamList[1] := 'BusNum';
ParamList[2] := 'GenID';
ParamList[3] := 'GenMW';

// ValueList is setup with MW values for generators at buses 1 and 2
ValueList := VarArrayCreate([1,2], varOleStr);
For k := 1 to 2 do begin
IndValueList := VarArrayCreate([1,3], varOleStr);
IndValueList[1] := k;
IndValueList[2] := '1';
IndValueList[3] := k*10;
ValueList[k] := IndValueList;
End;

// Make the ChangeParametersMultipleElement call
Output = SimAuto.ChangeParametersMultipleElement('Sim_Solution_Options', _
ParamList, ValueList)

```

Microsoft® Visual Basic for Applications

```

' Set ParamList up to modify the MW output of generators
Dim ParamList As Variant
ParamList = Array("BusNum", "GenID", "GenMW")

' ValueList is setup with MW values for generators at buses 1 and 2
Dim ValueList(2) As Variant
For k = 0 to 1
Dim IndValueList As Variant
IndValueList = Array(k+1,"1",(k+1)*10)
ValueList(k) = IndValueList
Next

' Make the ChangeParametersMultipleElement call

```

```
Output = SimAuto.ChangeParametersMultipleElement("Sim_Solution_Options", _  
ParamList, ValueList)
```

ChangeParametersMultipleElementFlatInput Function

The ChangeParametersMultipleElementFlatInput function allows you to set parameters for multiple objects of the same type in a case loaded into the [Simulator Automation Server](#). This function is very similar to the [ChangeParametersMultipleElement](#), but uses a single dimensioned array of values as input instead of a multi-dimensioned array of arrays. The advantage of this function is that it is much faster to change multiple elements with a single function call than it is to repeatedly call [ChangeParametersSingleElement](#) multiple times. An additional advantage over [ChangeParametersMultipleElement](#) is that you can still take advantage of the speed improvement, even if the programming language you are using does not support multi-dimensioned arrays.

Function Prototype

ChangeParametersMultipleElement(ObjectType, ParamList, NoOfObjects, ValueList)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

NoOfObjects You must pass an integer number of devices that are passing values for. SimAuto will automatically check that the number of parameters for each device (counted from ParamList) and the number of objects integer correspond to the number of values in value list (counted from ValueList.)

ValueList : Variant A variant array storing a list of variants. Value list can be an array with many values, as it is a single dimensioned array of all values for all devices that are being changed. The structure of the ValueList array is such that all of the parameters for the first object are listed first, then all parameters for the second object, and so on. The parameters must be in the same order as given in ParamList. In other words, your array would look like:

```
ValueList = Array(Obj1Param1, Obj1Param2, Obj2Param1, Obj2Param2, Obj3Param1, ...,
ObjNParam1, ObjNParam2)
```

Output

ChangeParametersMultipleElementFlatInput only returns the first element in Output, the error string.

Notes

If the number of parameters given in ParamList multiplied by the number of objects passed does not equal the total number of values in ValueList, SimAuto will abort the function call.

ChangeParametersMultipleElementFlatInput Sample Code

Borland® Delphi

```

Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the MW output of generators
ParamList := VarArrayCreate([1,3], varOleStr);
ParamList[1] := 'BusNum';
ParamList[2] := 'GenID';
ParamList[3] := 'GenMW';

// ValueList is setup with MW values for generators at buses 1 and 2
NumObjects := 2;
NumFields := NumObjects * 3;
ValueList := VarArrayCreate([1,NumFields], varOleStr);
For k := 0 to 1 do begin
ValueList[3*k+1] := k;
ValueList[3*k+2] := '1';
ValueList[3*k+3] := k*10;
End;

// Make the ChangeParametersMultipleElementFlatInput call
Output = SimAuto.ChangeParametersMultipleElementFlatInput('Sim_Solution_Options', _
ParamList, NumObjects, ValueList)

```

Microsoft® Visual Basic for Applications

```

' Set ParamList up to modify the MW output of generators
Dim ParamList As Variant
ParamList = Array("BusNum", "GenID", "GenMW")

' ValueList is setup with MW values for generators at buses 1 and 2
NumObjects = 2
NumFields = NumObjects * 3
Dim ValueList(NumObjects) As Variant
For k = 0 to 1
ValueList(3*k+1) = k
ValueList[3*k+2] := '1';
ValueList[3*k+3] := k*10;
Next

```

```
' Make the ChangeParametersMultipleElementFlatInput call  
Output = SimAuto.ChangeParametersMultipleElementFlatInput("Sim_Solution_Options", _  
ParamList, NumObjects, ValueList)
```

CloseCase Function

The CloseCase function is used to close a load flow case loaded in the [Simulator Automation Server](#). This function should be called at some point after the [OpenCase](#) function.

Function Prototype

CloseCase()

Parameter Definitions

No parameters are passed.

Output

CloseCase returns only one element in Output—any errors which may have occurred when attempting to close the case.

CloseCase Function: Sample Code

Borland® Delphi

Output := SimAuto.CloseCase();

Microsoft® Visual Basic for Applications

Output = SimAuto.CloseCase()

Matlab®

Output = SimAuto.CloseCase

GetFieldList Function

The GetFieldList function is used to find all fields contained within a given object type.

Function Prototype

GetFieldList(ObjectType)

Parameter Definitions

ObjectType : String The type of object for which the fields are requested.

Output

GetFieldList returns two elements of the Output array. The first element, as with the other functions, returns any errors that might have occurred. The second element of the Output array contains an $n \times 4$ array of fields. The layout of this array is virtually identical to the output obtained by going to **Help > Export Case Object Fields** from the main menu of Simulator. The first column, corresponding to the $(n,0)$ column in the field array, specifies which fields are key fields for the object. The second column, $(n,1)$, contains the internal name of the field. The third column, $(n,2)$, contains the type of data stored in the string (e.g. String, Integer, Real). The fourth column, $(n,3)$, contains the display-friendly name of the field.

GetFieldList Function: Sample Code

Microsoft® Visual Basic for Applications

```
Dim objecttype As String
```

```
' Object type to obtain  
objecttype = "branch"
```

```
' Make the GetField call  
Output = SimAuto.GetFieldList(objecttype)
```

Matlab®

```
% Object type to obtain  
objecttype = 'branch';
```

```
% Make the GetField call  
Output = SimAuto.GetFieldList(objecttype);
```

GetParametersSingleElement Function

The GetParametersSingleElement function is used to request the values of specified fields for a particular object in the load flow case. For returning field values for multiple objects, you can use a loop to make repeated calls to the GetParametersSingleElement function, and pass the object and desired field information for each object. This function is identical in setup to the [ChangeParameters](#) function, with the exception that the Values array will be updated with the values for the field variables defined in ParamList.

Function Prototype

GetParametersSingleElement(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

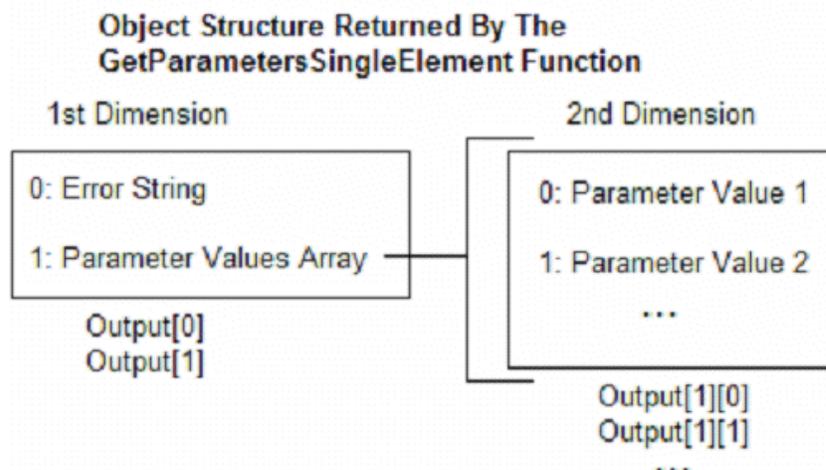
ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. Values must be passed for the [key field](#) variables in ParamList, in the same array position. The remaining field positions in the Values array should be set to zero.

Output

GetParametersSingleElement returns both the first element in Output—containing any errors occurring during execution of the function—and a second element in Output. The second element returned in the Output structure is a one dimensional array containing the values corresponding to the fields specified in ParamList.

The Output structure of GetParametersSingleElement is shown in the following figure.



GetParametersSingleElement Function: Sample Code Borland® Delphi

```

// This example retrieves some parameters for bus 2 of the loaded
// case, using the GetParametersSingleElement function, as well as
// the old GetParameters function

procedure TMainForm.RunGPSEClick(Sender: TObject);
var
  Output : OLEVariant;
  FieldBusArray, ValueBusArray : OLEVariant;
  i : Integer;
begin
  // Declares fields array to be sent to Excel
  FieldBusArray := VarArrayCreate([1,5], varOleStr);
  FieldBusArray[1] := 'pwBusNum';
  FieldBusArray[2] := 'pwBusname';
  FieldBusArray[3] := 'pwBusKVolt';
  FieldBusArray[4] := 'pwBusPUVolt';
  FieldBusArray[5] := 'pwBusAngle';
  ValueBusArray := varArrayCreate([1,5],varOleStr);
  ValueBusArray[1] := 2; // To get parameters for bus 2
  ValueBusArray[2] := 0;
  ValueBusArray[3] := 0;
  ValueBusArray[4] := 0;
  ValueBusArray[5] := 0;

  // Gets parameters with GetParametersSingleElement function
  Output := SimAuto.GetParametersSingleElement('bus', FieldBusArray, ValueBusArray);
  if (string(Output[0]) <> '') then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParametersSingleElement ==');
      Memo1.Lines.Add('Value : Output[1][i]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
        Memo1.Lines.Add(FieldBusArray[i] + ' : ' + string(Output[1][i]));
      end;
      Memo1.Lines.Add("");
    end;
end;

```

```
end;
```

```
// Gets parameters with old function GetParameters
```

```
Output := SimAuto.GetParameters('bus', FieldBusArray, ValueBusArray);
```

```
if (string(Output[0]) <> "") then
```

```
  StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
```

```
else
```

```
  begin
```

```
    StatusBar1.Panels[1].Text := 'Parameters got.';
```

```
    Memo1.Lines.Add('== GetParameters ==');
```

```
    Memo1.Lines.Add('Value : Output[1][i]');
```

```
    for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
```

```
      Memo1.Lines.Add(FieldBusArray[i] + ' : ' + string(Output[1][i]));
```

```
    end;
```

```
    Memo1.Lines.Add("");
```

```
  end;
```

```
end;
```

GetParametersSingleElement Function: Sample Code Matlab®

```

% This example loads all buses in the case, and then gets
% some parameters of the last bus in the list

% validcase is a global variable – check case is open
if validcase
    % Gets all buses in the case
    output = simauto.ListOfDevices('bus,');
    if ~(strcmp(output{1},''))
        disp(output{1})
        validbusarray = false;
    else
        % Puts the buses in row vector busarray
        for i=size(output{2}{1},1):size(output{2}{1},2)
            busarray(i,1) = output{2}{1}(i);
        end
        disp('Successful ListOfDevices')
        disp(busarray)
        validbusarray = true;
    end
end

% validbusarray is a global variable – check buses are loaded
if validcase & validbusarray
    % Gets parameters for last bus of busarray
    fieldarray = {'pwbusnum' 'pwbusname' 'pwbusvolt' 'pwbusangle'};
    valuearray = [busarray(size(busarray,1)) '0' '0' '0'];
    valuelist = num2cell(valuearray);
    output = simauto.GetParametersSingleElement('bus',fieldarray,valuelist);
    if ~(strcmp(output{1},''))
        disp(output{1})
    else
        % Puts the buses in row vector busparam
        paramlist = transpose(output{2});
        for i=size(paramlist,1):size(paramlist,2)
            busparam(i,1) = paramlist(i);
        end
        disp('Successful GetParameters for Bus')
    end
end

```

```
disp(fieldarray)
disp(busparam)
end
end
```

GetParametersSingleElement Function: Sample Code Microsoft® Visual Basic for Excel

```

Private Sub btnGetParametersSingleElement_Click()
Dim objtype, filter As String
Dim xlWB As Excel.Workbook
Set xlApp = Excel.Application

' Checks connection and open case
' SimAuto and caseopen are global variables
If Not SimAuto Is Nothing And caseopen Then
objtype = "bus"
Dim fieldArray As Variant
fieldArray = Array("pwBusNum", "pwBusName", "pwBusKVolt", _
"pwBusPUVolt", "pwBusAngle")
Dim ValueArray As Variant
ValueArray = Array(1, 0, 0, 0, 0)
output = SimAuto.GetParameters(objtype, fieldArray, ValueArray)
If output(0) <> "" Then
DisplayErrorMessage output(0)
Else
DisplayMessage "Successful GetParametersSingleElement"
' Prepares additional worksheet
Set xlWB = xlApp.Workbooks.Add
' Copies list of devices in worksheet
With xlWB
Sheets("sheet2").Activate
Sheets("sheet2").Name = "GetParametersSingleElement"
With Sheets("GetParametersSingleElement")
Dim i As Integer
Range(Cells(1, 5), Cells(200, 7)).Clear
Cells(1, 1) = "List of Devices for " + objtype + ":"
' Setup fields as subheader
For i = LBound(fieldArray) To UBound(fieldArray)
Cells(2, i + 1) = fieldArray(i)
Next i
' Determine number of fields retrieved
Dim lowfld, highfld As Integer
lowfld = LBound(output(1), 1)
highfld = UBound(output(1), 1)

```

```
DisplayMessage "Number of Fields: " + Str(lowfld) + Str(highfld)
For i = lowfld To highfld
Cells(j + 3, i + 1) = output(1)(i)
Next i
End With
End With
End If
End If
End Sub
```

GetParametersMultipleElement Function

The GetParametersMultipleElement function is used to request the values of specified fields for a set of objects in the load flow case. The function can return values for all devices of a particular type, or can return values for only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case.

Function Prototype

GetParametersMultipleElement(ObjectType, ParamList, FilterName)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

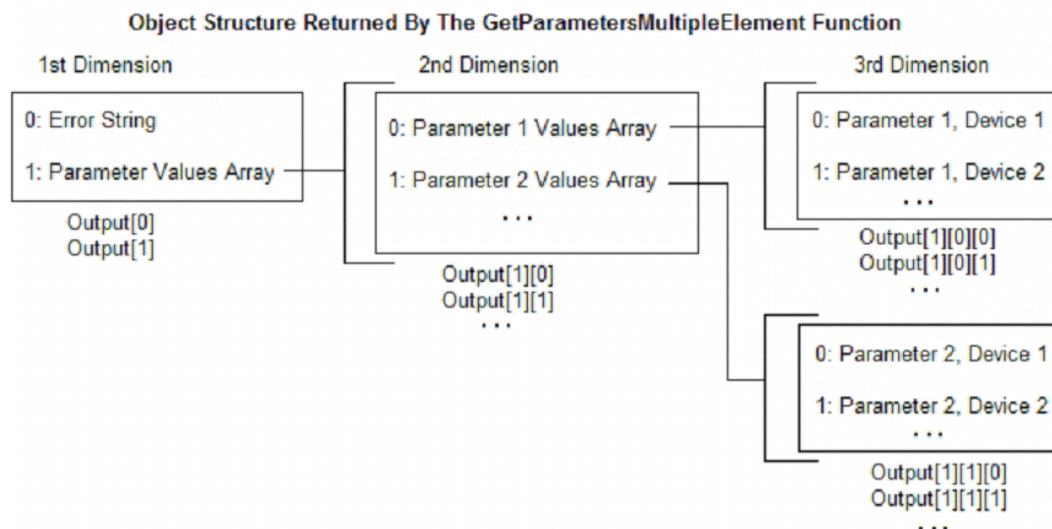
ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator.

FilterName : String The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, the server will default to returning all objects in the case of type ObjType.

Output

GetParametersMultipleElement returns a set of nested arrays containing the parameter values for the device type requested. The number of arrays of values returned depends on the number of fields in ParamList.

The Output structure of GetParametersMultipleElement is shown in the following figure.



As you can see, to access the first parameter value for the first device, `Output[1][0][0]` would be the correct array index. For example, the bus number for the first bus would be stored at `Output[1][0][0]` after calling

`Output = GetParametersMultipleElement('Bus', fieldarray, '')`, and assuming that we have `fieldarray = Array(pwBusnum, pwBusName)`.

GetParametersMultipleElement Sample Code Borland® Delphi

```
// This example retrieves some parameters for all buses of the
// loaded case, using the GetParametersMultipleElement function

procedure TMainForm.RunGPMEClick(Sender: TObject);
var
  FieldBusArray : OLEVariant;
  i,j : Integer;
begin
  // Declares fields array to be sent to Excel
  FieldBusArray := VarArrayCreate([1,5], varOleStr);
  FieldBusArray[1] := 'pwBusNum';
  FieldBusArray[2] := 'pwBusname';
  FieldBusArray[3] := 'pwBusKVolt';
  FieldBusArray[4] := 'pwBusPUVolt';
  FieldBusArray[5] := 'pwBusAngle';

  // Gets parameters with Multiple Element function
  Output := SimAuto.GetParametersMultipleElement('bus', FieldBusArray, "");
  if (string(Output[0]) <> "") then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParametersMultipleElement ==');
      Memo1.Lines.Add('Value : Output[1][i][j]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
        for j := VarArrayLowBound(Output[1][i],1) to VarArrayHighBound(Output[1][i],1)
        do begin
          Memo1.Lines.Add(FieldBusArray[i] + '(' + IntToStr(j) + ') : ' +
            string(Output[1][i][j]));
        end;
      end;
      Memo1.Lines.Add("");
    end;
end;
```

GetParametersMultipleElement Function: Sample Code Matlab®

```

% This example loads all buses in the case, and then gets
% some parameters of such buses

% validcase is a global variable – check case is open
if validcase
    % Gets all buses in the case
    output = simauto.ListOfDevices('bus', '');
    if ~(strcmp(output{1},''))
        disp(output{1})
        validbusarray = false;
    else
        % Puts the buses in row vector busarray
        for i=size(output{2}{1},1):size(output{2}{1},2)
            busarray(i,1) = output{2}{1}(i);
        end
        disp('Successful ListOfDevices')
        disp(busarray)
        validbusarray = true;
    end
end

% validbusarray is a global variable – check buses are loaded
if validcase & validbusarray
    % Gets parameters for all buses
    fieldarray = {'pwbusnum' 'pwbusname' 'pwbusvolt' 'pwbusangle'};
    output = simauto.GetParametersMultipleElement('bus', fieldarray, ' ');
    if ~(strcmp(output{1},''))
        disp(output{1})
    else
        % Puts the buses in matrix busesparam
        paramlist = transpose(output{2});
        for i=size(paramlist,1):size(paramlist,2)
            for j=size(paramlist{i},2):size(paramlist{i},1)
                busesparam(j,i) = paramlist{i}(j);
            end
        end
    end
    disp('Successful GetParametersMultipleElement')
end

```

```
disp(fieldarray)
disp(busesparam)
end
end
```

GetParametersMultipleElement Function: Sample Code Microsoft® Visual Basic for Excel

```

Private Sub btnGetParametersMultiple_Click()
Dim objtype, filter As String
Dim xlWB As Excel.Workbook
Set xlApp = Excel.Application

' Checks connection and open case
' SimAuto and caseopen are global variables
If Not SimAuto Is Nothing And caseopen Then
objtype = "bus"
filter = ""
Dim fieldArray As Variant
fieldArray = Array("pwBusNum", "pwBusName", "pwBusKVolt", _
"pwBusPUVolt", "pwBusAngle")
output = SimAuto.GetParametersMultipleElement(objtype, fieldArray, filter)
If output(0) <> "" Then
DisplayErrorMessage output(0)
Else
DisplayMessage "Successful GetParametersSingleElement"
' Prepares additional worksheet
Set xlWB = xlApp.Workbooks.Add
' Copies list of devices in worksheet
With xlWB
Sheets("sheet2").Activate
Sheets("sheet2").Name = "GetParametersSingleElement"
With Sheets("GetParametersSingleElement")
Dim i, j As Integer
Range(Cells(1, 5), Cells(200, 7)).Clear
Cells(1, 1) = "List of Devices for " + objtype + ":"
' Setup fields as subheader
For i = LBound(fieldArray) To UBound(fieldArray)
Cells(2, i + 1) = fieldArray(i)
Next i
' Determine number of fields retrieved
Dim lowfld, highfld As Integer
lowfld = LBound(output(1), 1)
highfld = UBound(output(1), 1)
' Determine number of objects retrieved

```

```
Dim lowobj, highobj As Integer
lowobj = LBound(output(1)(lowkeyf), 1)
highobj = UBound(output(1)(lowkeyf), 1)
DisplayMessage "Number of Fields: " + Str(lowfld) + Str(highfld)
DisplayMessage "Number of objects: " + Str(lowobj) + Str(highobj)
For i = lowfld To highfld
For j = lowobj To highobj
Cells(j + 3, i + 1) = output(1)(i)(j)
Next j
Next i
End With
End With
End If
End If
End Sub
```

GetParametersMultipleElementFlatOutput Function

This function operates the same as the [GetParametersMultipleElement](#) function, only with one notable difference. The values returned as the output of the function are returned in a single-dimensional vector array, instead of the multi-dimensional array as described in the [GetParametersMultipleElement](#) topic. The function returns the parameter values for the device type requested, and the number of values returned depends on the ParamList and any [advanced filter](#) that impacts the number of devices returned.

The format of the output array is the following:

```
[errorString, NumberOfObjectsReturned, NumberOfFieldsPerObject, Ob1Fld1, Ob1Fld2, ...,  
Ob(n)Fld(m-1), Ob(n)Fld(m)]
```

The data is thus returned in a single dimension array, where the parameters NumberOfObjectsReturned and NumberOfFieldsPerObject tell you how the rest of the array is populated. Following the NumberOfObjectsReturned parameter is the start of the data. The data is listed as all fields for object 1, then all fields for object 2, and so on. You can parse the array using the NumberOf... parameters for objects and fields.

GetParameters Function

This function is maintained in versions of Simulator later than version 9 for compatibility with Simulator version 9. This function is replaced by [GetParametersSingleElement](#). See also [GetParametersMultipleElement](#).

ListOfDevices Function

The ListOfDevices function is used to request a list of objects and their [key fields](#) from the [Simulator Automation Server](#). The function can return all devices of a particular type, or can return only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case. This function is best used in conjunction with a looping procedure and the [ChangeParameters](#) or [GetParametersSingleElement](#) functions to process a group of devices.

Function Prototype

ListOfDevices(ObjType, filterName)

Parameter Definitions

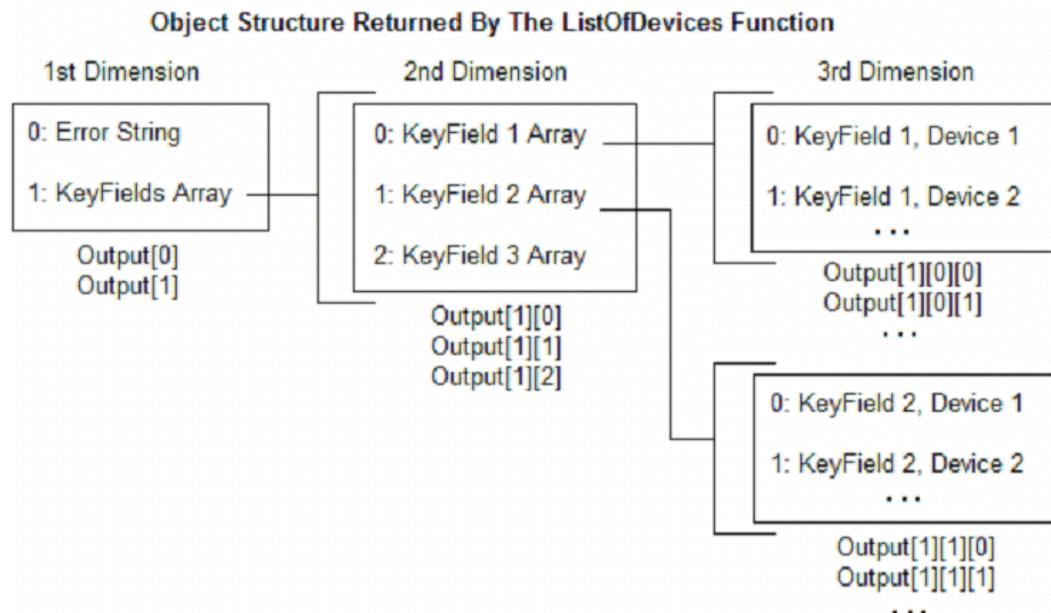
ObjType : String The type of object for which you are acquiring the list of devices.

FilterName : String The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If the filter cannot be found, the server will default to returning all objects in the case of type ObjType.

Output

ListOfDevices returns a set of nested arrays containing the [key field](#) values for the device type requested. The number of arrays of values returned depends on the object type selected. For instance, buses have only one key field (the bus number) so calling ListOfDevices for buses will return only one array of values—the bus numbers. On the other hand, calling ListOfDevices for branches will return three arrays of values—the "From" bus, "To" bus, and ID—for each branch in the case meeting the specified filter.

The arrays containing the key field values for each device are arranged as shown in the following figure.



As you can see, to access the first key field value for the first device, `Output[1][0][0]` would be the correct array index. For example, the bus number (which is the bus key field) for the first bus would be stored at `Output[1][0][0]` after calling `Output = ListOfDevices('Bus', '')`.

One unique limitation of the `ListOfDevices` function from other `SimAuto` functions is that this is the only function that returns the output as strongly typed variables. The bus numbers are always returned as Long Integers, and the Circuit ID values are returned as strings. This was actually an oversight during the design of `SimAuto`. In all other `SimAuto` functions, the values are returned as Variant types, with each value within the variant being a string. This was the intended operation for this function as well. Since the Automation Server interface was released with the errant inclusion of the `ListOfDevices` function, it could not be modified. Therefore, another function, `ListOfDevicesAsVariantStrings`, has been created. This function returns all values in variant variables, with each as a string within the variant type.

ListOfDevices Function: Sample Code for Borland® Delphi

Sample Code

```
// Runs Available Transfer Capability Routine
// Executes ATC Calculations among all areas
// and sends results to Excel
procedure TMainForm.RunATCClick(Sender: TObject);
var
  i, j, LowB, HighB : Integer;
  ValuesAreaArray : OLEVariant;
begin
  // Obtain all the areas
  Output := SimAuto.ListOfDevices('area', "");
  if (string(Output[0]) <> "") then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      ValuesAreaArray := Output[1][0];
      LowB := VarArrayLowBound(ValuesAreaArray, 1);
      HighB := VarArrayHighBound(ValuesAreaArray, 1);
      // Executes loop
      for i := LowB to HighB do
        for j := LowB to HighB do begin
          if (i <> j) then begin
            // Runs ATC calculations
            OutputATC := SimAuto.RunScriptCommand('entermode(atc); ' +
            'atcdetermine([Area ' + IntToStr(ValuesAreaArray[i]) +
            '], [Area ' + IntToStr(ValuesAreaArray[j]) + '])');
            if (string(OutputATC[0]) <> "") then
              StatusBar1.Panels[1].Text := 'Error: ' + string(OutputATC[0])
            else begin
              // Sends ATC results to Excel
              Output := SimAuto.SendToExcel('transferlimiter', "", 'all');
              if (string(Output[0]) <> "") then
                StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
              else
                StatusBar1.Panels[1].Text := 'ATC calculations done.';
            end;
          end;
        end;
    end;
end;
```

```
end;  
end;  
end;
```

ListOfDevices Function: Sample Code for Matlab®

Sample Code

```

%A list of branches is desired, without using any filter
DeviceType = 'Branch';
FilterName = "";
%Execute the ListOfDevices command
Output = SimAuto.ListOfDevices(DeviceType,FilterName);
%If the first cell in SimAutoOutput ~= "", then that means an error
%occurred.
if ~(strcmp(SimAutoOutput{1},""))
    disp(SimAutoOutput{1})
else
    %Otherwise, no errors. Display the branch information.
    disp('ListOfDevices successful')

    %Devicelist1 contains the From: bus number
    %Devicelist2 contains the To: bus number
    %Devicelist3 contains the Bus identifier
    devicelist1 = double(transpose(SimAutoOutput{2}{1}));
    devicelist2 = double(transpose(SimAutoOutput{2}{2}));
    devicelist3 = SimAutoOutput{2}{3};

    %If the device list is greater than 25, don't bother attempting to
    %display it on the screen.
    if (size(devicelist1,1) > 25)
        disp('Device list exceeds 25; use "devicelist" to manage list of devices')
    else
        %Otherwise, display the branches' information.
        disp(DeviceType)
        disp('From/To/Identifier')
        for counter = 1:size(devicelist1,1)
            %num2str converts the numbers within devicelist1 and
            %devicelist2 to strings for output with disp(). the char()
            %function is called on devicelist3's members because the
            %SimAuto object returns a character array (as opposed to a
            %properly Matlab-format string) and this array must be
            %converted to a Matlab-format string.
            disp([num2str(devicelist1(counter)) ' ' ...

```

```
num2str(devicelist2(counter)) ' ' ...  
char(devicelist3(counter))])  
end  
end  
end
```

ListOfDevices Function: Sample Code for Microsoft® Visual Basic for Excel

Sample Code

```
Private Sub DisplayMessage(ByVal SentText As String)
    TextBox.Text = TextBox.Text + SentText + vbCrLf + vbCrLf
End Sub
```

```
Private Sub btnListOfDevices_Click()
    Dim objtype, filter As String
    Dim xlWB As Excel.Workbook
    Set xlApp = Excel.Application

    ' Checks connection and open case
    ' SimAuto and caseopen are global variables
    If Not SimAuto Is Nothing And caseopen Then
        objtype = "branch"
        filter = ""
        output = SimAuto.ListOfDevices(objtype, filter)
        If output(0) <> "" Then
            DisplayMessage output(0)
        Else
            DisplayMessage "Successful List Of Devices"
        ' Prepares additional worksheet
        Set xlWB = xlApp.Workbooks.Add
        ' Copies list of devices in worksheet
        With xlWB
            Sheets("sheet1").Activate
            Sheets("sheet1").Name = "ListOfDevices"
            With Sheets("ListOfDevices")
                Dim i, j As Integer
                Range(Cells(1, 5), Cells(200, 7)).Clear
                Cells(1, 1) = "List of Devices for " + objtype + ":"
                Cells(2, 1) = "From Bus Num"
                Cells(2, 2) = "To Bus Num"
                Cells(2, 3) = "ID"
            ' Determine number of key fields retrieved
            Dim lowkeyf, highkeyf As Integer
            lowkeyf = LBound(output(1), 1)
            highkeyf = UBound(output(1), 1)
        End With
    End If
End Sub
```

```
DisplayMessage "Number of Key Fields: " + Str(lowkeyf) + Str(highkeyf)
' Determine number of objects retrieved
Dim lowobj, highobj As Integer
lowobj = LBound(output(1)(lowkeyf), 1)
highobj = UBound(output(1)(lowkeyf), 1)
DisplayMessage "Number of objects: " + Str(lowobj) + Str(highobj)
For i = lowkeyf To highkeyf
For j = lowobj To highobj
Cells(j + 3, i + 1) = output(1)(i)(j)
Next j
Next i
End With
End With
End If
End If
End Sub
```

ListOfDevicesAsVariantStrings Function

This function operates the same as the [ListOfDevices](#) function, only with one notable difference. The values returned as the output of the function are returned as Variants of type String. The ListOfDevices function was errantly released returning the values strongly typed as Integers and Strings directly, whereas all other SimAuto functions returned data as Variants of type String. This function was added to also return the data in the same manner. This solved some compatibility issues with some software languages.

ListOfDevicesFlatOutput Function

This function operates the same as the [ListOfDevices](#) function, only with one notable difference. The values returned as the output of the function are returned in a single-dimensional vector array, instead of the multi-dimensional array as described in the ListOfDevices topic. The function returns the key field values for the device, typically in the order of bus number 1, bus number 2 (where applicable), and circuit identifier (where applicable). These are the most common key fields, but some object types do have other key fields as well.

The format of the output array is the following:

```
[errorString, NumberOfObjectsReturned, NumberOfFieldsPerObject, Ob1Fld1, Ob1Fld2, ...,  
Ob(n)Fld(m-1), Ob(n)Fld(m)]
```

The data is thus returned in a single dimension array, where the parameters NumberOfObjectsReturned and NumberOfFieldsPerObject tell you how the rest of the array is populated. Following the NumberOfObjectsReturned parameter is the start of the data. The data is listed as all fields for object 1, then all fields for object 2, and so on. You can parse the array using the NumberOf... parameters for objects and fields.

LoadState Function

LoadState is used to load the system state previously saved with the [SaveState](#) function. Note that LoadState will not properly function if the system topology has changed due to the addition or removal of the system elements.

Function Prototype

LoadState()

Parameter Definitions

No parameters are passed.

Output

LoadState returns only one element in Output—any errors which may have occurred when attempting to execute the function.

LoadState Function: Sample Code

Microsoft® Visual Basic for Applications

```
' Make the LoadState call  
Output = SimAuto.LoadState()
```

Matlab®

```
% Make the LoadState call  
Output = SimAuto.LoadState();
```

OpenCase Function

The OpenCase function will load a PowerWorld® Simulator load flow file into the [Simulator Automation Server](#). This is equivalent to opening a file using the **File > Open Case** menu option in Simulator.

Function Prototype

OpenCase(FileName)

Parameter Definitions

FileName : String The name of the PowerWorld® Simulator case file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

Output

OpenCase returns only one element in Output—if the file cannot be found or an error occurs while reading the file.

OpenCase Function: Sample Code

Borland® Delphi

```
Output := SimAuto.OpenCase('c:\simauto\examples\b7opf.pwb');
if (string(Output[0]) <> "") then
  StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0]);
else
begin
  StatusBar1.Panels[1].Text := 'Open Case successful.';
  // Perform activities with opened case
end;
```

Microsoft® Visual Basic for Applications

```
Output = SimAuto.OpenCase("c:\simauto\examples\b7opf.pwb")
If output(0) <> "" Then
  MsgBox(output(0))
Else
  ' Perform activities with the opened case
End If
```

Matlab®

```
Output = SimAuto.OpenCase('c:\simauto\examples\b7opf.pwb')
%If the first cell in Output ~= "", then that means an error
%occurred.
if ~(strcmp(Output{1}, ""))
  disp(Output{1})
else
  %Otherwise, no errors. Perform activities.
  disp('Open Case successful')
end
```

ProcessAuxFile Function

The ProcessAuxFile function will load a PowerWorld® Auxiliary file into the [Simulator Automation Server](#). This allows you to create a text file (conforming to the PowerWorld® Auxiliary file format) that can list a set of data changes and other information for making batch changes in Simulator

Function Prototype

ProcessAuxFile(FileName)

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

Output

ProcessAuxFile returns only one element in Output—any errors which may have occurred when attempting to load the file.

ProcessAuxFile Function: Sample Code

Microsoft® Visual Basic for Applications

Dim filename As String

' Setup name of aux file to run
filename = "c:\b7opf_ctglist.aux"

' Make the processAuxFile call
Output = SimAuto.ProcessAuxFile(filename)

Matlab®

% Setup name of aux file to run
filename = 'c:\b7opf_ctglist.aux';

% Make the processAuxFile call
Output = SimAuto.ProcessAuxFile(filename);

RunScriptCommand Function

The RunScriptCommand function is used to execute a list of [script statements](#). The [script actions](#) are those included in the script sections of the [Auxiliary Files](#). If an error occurs trying to run a script command, an error will be returned through EString.

Function Prototype

RunScriptCommand(Statements)

Parameter Definitions

Statements : String The block of script actions to be executed. Each script statement must end in a semicolon. The block of script actions should **not** be enclosed in curly braces.

Output

RunScriptCommand returns only one element in Output—any errors which may have occurred when attempting to load or run the auxiliary file.

RunScriptCommand Function: Sample Code

Microsoft® Visual Basic for Applications

```
Dim scriptcommand As String
```

```
' Set script command to cause Simulator to enter Run Mode  
scriptcommand = "EnterMode(RUN)"
```

```
' Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);
```

```
' Set script command to cause Simulator to perform a single,  
' standard solution  
scriptcommand = "SolvePowerFlow(RECTNEWT)"
```

```
' Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);
```

Matlab®

```
% Set script command to cause Simulator to enter Run Mode  
scriptcommand = 'EnterMode(RUN)';
```

```
% Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);
```

```
% Set script command to cause Simulator to perform a single,  
% standard solution  
scriptcommand = 'SolvePowerFlow(RECTNEWT)';
```

```
% Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);
```

SaveCase Function

The SaveCase function is used to save a case previously loaded in the [Simulator Automation Server](#) using the [OpenCase](#) function. The function allows you to specify a file name and a format for the save file.

Function Prototype

SaveCase(FileName, EString, FileType, Overwrite)

Parameter Definitions

FileName : String The name of the file you wish to save as, including file path.

FileType : String A string indicating the format of the written case file. An empty string will return an error. The following list is the currently supported list of string identifiers and the file types they represent.

"PTI23" PTI version 23 (raw)

"PTI24" PTI version 24 (raw)

"PTI25" PTI version 25 (raw)

"PTI26" PTI version 26 (raw)

"PTI27" PTI version 27/28 (raw)

"PTI29" PTI version 29 (raw)

"PTI30" PTI version 30 (raw)

"PTI31" PTI version 31 (raw)

"GE" GE PSLF (epc)

"IEEE" IEEE common format (cf)

"PWB70" PowerWorld Binary version 7.0 (pwb)

"PWB" PowerWorld Binary (most recent) (pwb)

Overwrite : Boolean A Boolean value which indicates whether to overwrite a file if FileName already exists. If Overwrite is set to false and the file specified by FileName already exists, SaveCase will return an error message and do nothing to the file.

Output

SaveCase returns only one element in Output—any errors which may have occurred when attempting to save the case.

SaveCase Function: Sample Code

Microsoft® Visual Basic for Applications

' Save the case as a PWB file

```
Output = SimAuto.SaveCase("c:\b7opfcopy.pwb", "PWB", true)
```

' Save the case as a PTI file

```
Output = SimAuto.SaveCase("c:\b7opfcopy.raw", "PTI", true)
```

Matlab®

% Setup name of PWB file to write

```
filenamepwb = 'c:\b7opfcopy.pwb';
```

% Setup name of PTI file to write

```
filenamepti = 'c:\b7opfcopy.raw';
```

% Make the SaveCase call for the PWB file

```
Output = SimAuto.SaveCase(filenamepwb, 'PWB', true);
```

% Make the SaveCase call for the PTI file

```
Output = SimAuto.SaveCase(filenamepti, 'PWB', true);
```

SaveState Function

SaveState is used to save the current state of the power system. This can be useful if you are interested in comparing various cases, much as the [Difference Flows](#) feature works in the Simulator application.

Function Prototype

SaveState()

Parameter Definitions

No parameters are passed.

Output

SaveState returns only one element in Output—any errors which may have occurred when attempting to execute the function.

SaveState Function: Sample Code

Microsoft® Visual Basic for Applications

```
' Make the SaveState call  
Output = SimAuto.SaveState()
```

Matlab®

```
% Make the SaveState call  
Output = SimAuto.SaveState();
```

SendToExcel Function

The SendToExcel function can be called to send data from the [Simulator Automation Server](#) to an Excel spreadsheet. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few [field types](#) as desired that are supported by the type of object. The first time this function is called, a new instance of Excel will be started, and the data requested will be pasted to a new sheet. For each subsequent call of this function, the requested data will be pasted to a new sheet within the same workbook, until the workbook is closed.

Function Prototype

SendToExcel(ObjectType, FilterName, FieldList)

Parameter Definitions

ObjectType : String A string describing the type of object for which you are requesting data.

FilterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used.

FieldList : Variant This parameter must either be an array of fields for the given object or the string "all". As an array, FieldList contains an array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If, instead of an array of strings, the single string "all" is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Output

SendToExcel returns only one element in Output—any errors which may have occurred when attempting to execute the function.

SendToExcel Function: Sample Code

Microsoft® Visual Basic for Applications

```
Dim FieldList As Variant
```

```
' Setup fieldlist to send the bus number, gen id and gen agc to Excel
```

```
FieldList = Array("pwBusNum", "pwGenID", "pwGenAGCable")
```

```
' Make the SendToExcel call
```

```
' By specifying the parameter FieldList, only the three fields
```

```
' for each generator will be returned
```

```
Output = SimAuto.SendToExcel("gen", "", "FieldList")
```

```
' Sending the string "all" instead of a fieldlist array
```

```
' writes all predefined fields to the Excel spreadsheet
```

```
Output = SimAuto.SendToExcel("gen", "", "all")
```

Note: This function call will send the values of the fields in FieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

Matlab®

```
% Setup fieldlist to send the bus number, gen id and gen agc to Excel
```

```
fieldlist = {'pwBusNum' 'pwGenID' 'pwGenAGCable' };
```

```
% Make the SendToExcel call
```

```
Output = SimAuto.SendToExcel('gen', "", FieldList);
```

```
% Sending the string 'all' instead of a fieldlist array
```

```
% writes all predefined fields to the Excel spreadsheet
```

```
Output = SimAuto.SendToExcel('gen', "", 'all');
```

Note: This function call will send the values of the fields in FieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

WriteAuxFile Function

The WriteAuxFile function can be used to write data from the case in the [Simulator Automation Server](#) to a PowerWorld® Auxiliary file. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few field types as desired that are supported by the type of object. In addition, you can specify a new file name for each call to WriteAuxFile, or you can specify the same file name and append the data to the file.

Function Prototype

WriteAuxFile(FileName, FilterName, ObjectType, EString, ToAppend, FieldList)

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file you wish to save.

FilterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used.

ObjectType : String A string describing the type of object for which you are requesting data.

ToAppend : Boolean If you have given a file name of an auxiliary file that already exists, then the file will either be appended to or overwritten according to the setting of this parameter. If ToAppend is *False* and the file already exists, WriteAuxFile will return an error message and do nothing to the file.

FieldList : Variant This parameter must either be an array of fields for the given object or the string "all". As an array, FieldList contains an array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If, instead of an array of strings, the single string "all" is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Output

WriteAuxFile returns only one element in Output—any errors which may have occurred when attempting to execute the function.

WriteAuxFile Function: Sample Code

Microsoft® Visual Basic for Applications

```
Dim FieldList As Variant
```

```
Dim auxfilename As String
```

```
' Setup FieldList to send the bus number, gen id and gen agc
```

```
FieldList = Array("pwBusNum", "pwGenID", "pwGenAGCable")
```

```
' Aux file to write to
```

```
auxfilename = "c:\businfo.aux"
```

```
' Make the WriteAuxFile call
```

```
' By specifying the parameter FieldList, only the three fields
```

```
' for each generator will be returned
```

```
Output = SimAuto.WriteAuxFile(auxfilename, "", "gen", true, FieldList)
```

```
' Sending the string "all" instead of the FieldList array
```

```
' writes all predefined fields to the Excel spreadsheet
```

```
Output = SimAuto.SendToExcel(auxfilename, "", "gen", true, "all")
```

Note: This function call will send the values of the fields in FieldList to an auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

Matlab®

```
% Setup FieldList to send the bus number, gen id and gen agc
```

```
fieldlist = {'pwBusNum' 'pwGenID' 'pwGenAGCable' };
```

```
% Aux file to write to
```

```
auxfilename = 'c:\businfo.aux';
```

```
% Make the WriteAuxFile call
```

```
Output = SimAuto.WriteAuxFile(auxfilename, "", 'gen', true, FieldList);
```

```
% Sending the string 'all' instead of the FieldList array
```

```
% writes all predefined fields to the .aux file
```

```
Output = SimAuto.WriteAuxFile(auxfilename, "", 'gen', true, 'all');
```

Note: This function call will send the values of the fields in FieldList to an auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

Simulator Automation Server Properties

The following list of parameters is currently available once the SimulatorAuto object is set in your code. Check the help sections on these properties to see more detail on the particular property.

[ExcelApp](#)

[CurrentDir](#)

[ProcessID](#)

[RequestBuildDate](#)

ExcelApp Property

The [Simulator Automation Server](#) has the ability to send data from Simulator to an Excel spreadsheet using the [SendToExcel](#). By default, the Simulator Automation Server starts an instance of Excel the first time one of the above functions is called. Each subsequent call to these two functions will then send data to the same instance of Excel, until it is manually closed by the user. The ExcelApp property allows the user to gain access to the instance of Excel used by the Simulator Automation Server from within its own code. Thus the user can write code to manipulate the external instance of Excel. In addition, the ExcelApp property can be set by the user's code, meaning that the user can initialize an external instance of Excel from within their own code and set the ExcelApp property to their external instance of Excel. Simulator itself is limited to starting only one instance of Excel on its own, but with the ExcelApp property allowing you to set the instance of Excel that Simulator uses on the fly, you can generate multiple instances of Excel within your code, and handle setting the ExcelApp property of the Simulator Automation server to the desired Excel instance depending on the data you want to send to Excel.

The ExcelApp property is a variable of type Variant that returns the pointer to an object representing the external instance of Excel.

ExcelApp : Variant

To gain access to the external instance of Excel stored in the ExcelApp property, you first need to initialize a variable as an object. The following are a couple of examples in Borland Delphi and Microsoft Visual Basic.

Examples

Borland Delphi 5

- **Var MyExc : TObject**
- **MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **MyExc.Free***{Removes the connection to the external instance}*

Microsoft Visual Basic

- **Dim MyExc As Variant**
- **Set MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **Set MyExc = Nothing***{Removes the connection to the external instance}*

ExcelApp Property: Sample Code

Microsoft® Visual Basic for Applications

```
Dim ExcelObject As Variant
If Not (IsEmpty(SimAuto.ExcelApp)) Then
  Set ExcelObject = SimAuto.ExcelApp
  ExcelObject.DisplayAlerts = False
  ExcelObject.Quit
Else
  MsgBox("Attempted to obtain Excel COM object from " + _
  "Simulator, but there is not one open.")
End If
```

Matlab®

MATLAB is currently unable to handle COM object properties returning COM objects themselves, such as the ExcelApp property of Simulator.

CurrentDir Property

The CurrentDir property of the [Simulator Automation Server](#) allows you to retrieve or set the working directory for the currently running SimulatorAuto process. This is most useful if using relative filenames (e.g. "relativename.aux" versus "C:\Program Files\PowerWorld\Working\abosultename.aux") when specifying files.

CurrentDir Property: Sample Code

Microsoft® Visual Basic for Applications

' Display the current directory

```
MsgBox(SimAuto.CurrentDir)
```

' Set the current directory to c:\

```
SimAuto.CurrentDir = "c:\"
```

Matlab®

% Display the current directory

```
disp(SimAuto.CurrentDir)
```

% Set the current directory to c:\

```
SimAuto.CurrentDir = 'c:\';
```

ProcessID Property

The ProcessID property of the [Simulator Automation Server](#) allows you to retrieve the process ID of the currently running SimulatorAuto process, as can also be seen through the Task Manager in Windows® NT 4, 2000 and XP. This information can be useful if a forced shutdown of the SimulatorAuto object is needed, as all calls to the SimulatorAuto object are synchronous. This means the SimulatorAuto object will not be destroyed until all calls, no matter the time of execution, have completed.

ProcessID Property: Sample Code

Microsoft® Visual Basic for Applications

```
' Display the process ID  
MsgBox(SimAuto.ProcessID)
```

Matlab®

```
% Display the process ID  
disp(num2str(SimAuto.ProcessID))
```

RequestBuildDate Property

The RequestBuildDate property of the [Simulator Automation Server](#) allows you to retrieve the build date of the PowerWorld Simulator executable currently running with the SimulatorAuto process. The property returns an integer value that represents a date. This information is useful for verifying the release version of the executable.

PowerWorld Object Variables

The ability to access power system data for different objects through various [Simulator Automation Server Functions](#) is based on variables defined in Simulator that can be referred to as Object Field Variables. Each object (i.e. bus, generator, etc.) can have numerous fields associated with it. Each of these fields, in turn, has a variable associated with the field to enable access to the field for the purpose of acquiring or changing data. For example, the [GetParametersSingleElement](#) function has a parameter called ParamList, which is intended to store a list of Object Field Variables for a particular type of object. When the function is called, the Simulator Automation Server will return the values associated with each particular field variable for the type of object specified. These field variables allow for complete flexibility by the user in specifying as many or as few fields for a particular object when acquiring or changing data.

Examples of Field Variables

BusGenMW

BusNum

Simulator has literally hundreds of parameters spanning numerous types of device and option specifications. Rather than list all of the field variables and the value they represent in this help file, we have enabled Simulator to automatically generate a text file containing the field variables and a description of what value the variable represents. PowerWorld Corporation highly recommends that you examine this list. To generate this text file, run PowerWorld Simulator and access the **Help** menu. Choose the option **Export Case Object Fields...** The list of fields can either be saved to a text file or sent to Excel. The list will consist of the field variables, the type of variable (string, integer, etc.), and a description of the value the field variable represents, with key fields for different objects marked with an asterisk. The field variables will also be split into sections based on the type of object they are valid for. Note that the same field variable may be available for more than one object, but that the value represented by the field variable might vary for different objects.

Examples of Field Variables in Listing

BusGenMW Real 'Bus Gen MW'

* BusNum Integer 'Bus Number' (* denotes key field in text file)

ATC_MaxLimElements Integer 'Max # Limiting Elements'

The last note on the Object Field Variables is that some of the object field variables are reused for more than one value for an object. For example, a transmission line has "from" and "to" buses associated with the line. Rather than have separate field variables for values at each terminal bus, the same field variable is used for both, with a colon followed by a number appended to the variable to signify which bus the value represents. As an example, consider the field variable for bus per unit voltage, which is BusPUVolt. Since there are two buses per line, the "From" bus voltage would be represented as BusPUVolt:0, and the "To" bus voltage would be represented as BusPUVolt:1. The enumeration of the field variables always starts with 0 for the first instance. You may note that in the text file of field values that you don't see any field variables with :0 appended to them. Since we did not want you to have to always append the :0 on all field variables, the default for a field variable with no appended :# is 0. Thus you would only need to be concerned with appending the :# for field variables that require a number greater than 0.

Simulator Automation Server (version 9)

NOTE: The function calls for version 9 of SimAuto have become obsolete in version 10. The reason for the change was due to problems that arose with the structure of the functions in certain programming languages. The function structures have been modified for version 10 of Simulator. If you have code written that uses version 9 SimAuto function structures, it is fairly straightforward to convert the function calls from version 9 to version 10. Please see the updated section on the [Simulator Automation Server](#) for version 10. If you still have questions, please contact PowerWorld Corporation.

The PowerWorld® Automation Server is only available to customers who have purchased the SimAuto add-on for PowerWorld® Simulator. The PowerWorld Simulator Automation Server is intended for enabling a PowerWorld customer with the ability to access PowerWorld Simulator functionality from within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various different programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, and Microsoft® Visual Basic, just to name a few. For more information on COM and Automation servers, see the help for Microsoft Windows.

The Automation Server of Simulator works very well in combination with Simulator [Script Commands](#) and [Auxiliary Files](#). It is beneficial to become familiar with these topics when considering using the Simulator Automation Server.

Installing Simulator Automation Server (version 9)

Installing the [Simulator Automation Server](#) requires no additional steps beyond installing PowerWorld Simulator as normal. When a version of PowerWorld Simulator containing the Simulator Automation Server is installed on your computer, the install program automatically adds the information needed by the Simulator Automation Server to the registry.

Including Simulator Automation Server Functions (version 9)

Before you can access the [functions](#) defined by the [Simulator Automation Server](#) when writing the code for your external program, you must first include the library of functions defined for the Simulator Automation Server. This kind of library is referred to as a Type Library, which describes the available functions in a manner that can be interpreted by different programming languages. Importing a Type Library from another program is usually fairly simple, but the procedure does vary depending on the programming tool you are using. Please see the help for your programming tool of choice on how to import a Type Library or COM functions from another program.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi 5

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In Delphi 5, choose Import Type Library... from the Project menu.
- In the list of libraries, search for and choose pwrworld Library.
- If pwrworld Library is not in the list, click Add. Find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click Open.
- You should see the class name TSimulatorAuto in the list of Class names.
- Click Install to include the PowerWorld Simulator Type Library.

Microsoft Visual Basic

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In VB, choose **References...** from the **Tools** menu. Find and choose pwrworld Library from the list of references.
- If pwrworld Library is not in the list, click Browse. Change the file type to *.exe, find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click Open.
- Click OK to install the Simulator Type Library reference.

Microsoft Visual C++

- Install the version of PowerWorld Simulator with the Simulation Automation Server included.
- Add **#import "...\powerworld.exe"** in your external program code, using the full path to the PowerWorld Simulator executable program.
- Add **using namespace pwrworld** in your external program code.

Connecting to Simulator Automation Server (version 9)

Once the Type Library or COM functions have been included in your programming environment, the [Simulator Automation Server](#) can be handled as any other object in your code. The method for assigning and connecting to the Simulator Automation Server can vary depending on the programming environment used, but the idea is basically the same. You define a variable in your program to point to the server object, which is called SimulatorAuto. If the Type Library was imported properly, you should have full access to the SimulatorAuto object and its defined [functions](#). Again, the procedure for creating the object and connecting to SimulatorAuto may vary for different programming languages. Check the help for your programming environment on connecting to COM or Automation servers.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi 5

- Add **pwrworld_TLB** to the **uses** section of your unit.
- Declare a variable globally or as part of another object: **A : ISimulatorAuto.**
- Initialize the variable: **A := nil.**
- To connect to the Simulator Automation Server, create the connection: **A := CoSimulatorAuto.create.**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction(parameters).**
- To close the connection to the Simulator Automation Server, remove the reference by again setting: **A := nil.**

Microsoft Visual Basic

- Declare a variable globally or as part of another object or function: **Public A As SimulatorAuto.**
- To connect to the Simulator Automation Server, create the connection: **Set A = New SimulatorAuto.**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction parameters.**
- To close the connection to the Simulator Automation Server, remove the reference: **Set A = Nothing.**

Microsoft Visual C++

- Declare a variable globally or as part of another object or function: **IsimulatorAutoPtr *A.**
- Declare a variable globally or as part of another object or function: **CLSID clsid.**
- Declare a variable globally or as part of another object or function: **HRESULT hr.**
- Obtain the class identifier (clsid) with the following command: **hr = CLSIDFromProgID(L"pwrworld.SimulatorAuto", &clsid).**
- Initialize variable A: **A = new IsimulatorAutoPtr.**
- To connect to the Simulator Automation Server, create the connection: **hr = A>CreateInstance(clsid, NULL, CLSCTX_SERVER).**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction(parameters).**

- To close the connection to the Simulator Automation Server, release the reference: **hr = A>Release()**.

Simulator Automation Server Properties (version 9)

The following list of parameters is currently available once the SimulatorAuto object is set in your code. Check the help sections on these properties to see more detail on the particular property.

[ExcelApp](#)

ExcelApp Property (version 9)

The [Simulator Automation Server](#) has the ability to send data from Simulator to an Excel spreadsheet using the [SendToExcel](#). By default, the Simulator Automation Server starts an instance of Excel the first time one of the above functions is called. Each subsequent call to these two functions will then send data to the same instance of Excel, until it is manually closed by the user. The ExcelApp property allows the user to gain access to the instance of Excel used by the Simulator Automation Server from within their own code. Thus the user can write code to manipulate the external instance of Excel. In addition, the ExcelApp property can be set by the user's code, meaning that the user can initialize an external instance of Excel from within their own code and set the ExcelApp property to their external instance of Excel. Simulator itself is limited to starting only one instance of Excel on its own, but with the ExcelApp property allowing you to set the instance of Excel that Simulator uses on the fly, you can generate multiple instances of Excel within your code, and handle setting the ExcelApp property of the Simulator Automation server to the desired Excel instance depending on the data you want to send to Excel.

The ExcelApp property is a variable of type Variant that returns the pointer to an object representing the external instance of Excel.

ExcelApp : Variant

To gain access to the external instance of Excel stored in the ExcelApp property, you first need to initialize a variable as an object.

The following are a couple of examples in Borland Delphi and Microsoft Visual Basic.

Examples

Borland Delphi 5

- **Var MyExc : TObject**
- **MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **MyExc.Free***{Removes the connection to the external instance}*

Microsoft Visual Basic

- **Dim MyExc As Object**
- **Set MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **Set MyExc = Nothing***{Removes the connection to the external instance}*

Simulator Automation Server Functions (version 9)

The following list of functions is currently available once the SimulatorAuto object is set in your code. Check the help sections on these functions to see more detail on the particular function.

[ChangeParameters](#)

[CloseCase](#)

[GetParameters](#)

[ListOfDevices](#)

[LoadContingencies](#)

[OpenCase](#)

[ProcessAuxFile](#)

[RunScriptCommand](#)

[SaveCase](#)

[SendToExcel](#)

[WriteAuxFile](#)

ChangeParameters Function (version 9)

The ChangeParameters function allows you to set a list of parameters for a specific object in a case loaded into the [Simulator Automation Server](#). In addition to changing parameters for objects, this function can also be used to set options for some of the Simulator tools, such as [ATC](#) and [OPF](#). This function is identical in setup to the [GetParameters](#) function, with the exception that the Values array must contain a value for each field variable given in the ParamList array.

ChangeParameters(tObjectType, ParamList, Values, EString)**Parameter Definitions**

TObjectType : String The type of object you are changing parameters for. No default.

ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. No Default.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified. No Default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

Example

ChangeParameters("gen", [pwBusNum, pwGenID, pwGenAGCable], [1, "1", "Yes"], EString)

This function call will change the AGC Status of bus number one, generator ID number 1, to "Yes", meaning the generator will be included in AGC if it's area is on AGC control.

CloseCase Function (version 9)

The CloseCase function is used to close a load flow case loaded in the [Simulator Automation Server](#). This function should be called at some point after the [OpenCase](#) function. An error will be returned through the EString parameter if an error occurred while trying to close the case.

CloseCase(EString)**Parameter Definitions**

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

GetParameters Function (version 9)

The GetParameters function is used to request the values of specified fields for a particular object in the load flow case. For returning field values for multiple objects, you can use a loop to make repeated calls to the GetParameters function, and pass the object and desired field information for each object. This function is identical in setup to the [ChangeParameters](#) function, with the exception that the Values array will be updated with the values for the field variables defined in ParamList.

GetParameters(tObjectType, ParamList, Values, EString)

Parameter Definitions

TObjectType : String The type of object you are changing parameters for. No default.

ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator. No Default.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. Values must be passed for the [key field](#) variables in ParamList, in the same array position. The remaining field positions in the Values array can initially be filled with either empty strings or zeroes, as the values will be replaced when the function is processed. No Default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

Example

GetParameters("gen", [pwBusNum, pwGenID, pwGenAGCable], ValArray = [1, "1", ""], EString)

This function call will return the AGC Status of bus number one, generator ID number 1, in the ValArray variable. Note that the generator bus number and ID had to be passed into the function, while the remaining field could be assigned an empty string. If the bus number and ID had not been passed, an error would have been returned since Simulator would not be able to identify a generator. With a valid bus number and ID for the generator, the ValArray variable would have returned an array of the following format: ValArray = [1, "1", "Yes"].

ListOfDevices Function (version 9)

The ListOfDevices function is used to request a list of objects and their [key fields](#) from the [Simulator Automation Server](#). The function can return all devices of a particular type, or can return only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case. This function is best used in conjunction with a looping procedure and the [ChangeParameters](#) or [GetParameters](#) functions to process a group of devices.

ListOfDevices(tObjType, EString, filterName, objList1, {objList2}, {objList3})

Parameter Definitions

TobjType : String The type of object for which you are acquiring the list of devices. No default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

FilterName : String The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If the filter cannot be found, the server will default to returning all objects in the case of type TobjType.

ObjList1 : Variant This parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the first [key field](#) for the devices of type TobjType. This parameter is required for all types of devices.

ObjList2 : Variant This parameter **may** be optional. For device types that only have one [key field](#), you can omit passing a variable for this parameter. If the device type requested has two or more key fields, then this parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the second key field for the devices of type TobjType.

ObjList3 : Variant This parameter **may** be optional. For device types that have one or two [key fields](#), you can omit passing a variable for this parameter. If the device type requested has three key fields, then this parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the third key field for the devices of type TobjType.

Example

ListOfDevices("gen", EString, "", objList1, objList2)

This function call will return a list of all generators in the load flow case. By passing an empty string as the filter name, Simulator will not use an advanced filter before returning the generator key fields. Since generators have two key fields, bus number and ID, it was required to pass two variant array variables to receive the key fields. Each generator in the case would have its bus number returned in objList1 and ID in objList2. The corresponding bus number and ID are stored in the same index position in the two arrays. If only one variant array, objList1, had been passed in this instance, an error would have been returned stating more arrays were necessary. If the third variant array, objList3, had been included, Simulator would have still returned the key fields in objList1 and objList2, and would have returned objList3 as an empty variant.

LoadContingencies Function (version 9)

The LoadContingencies function can be used to read a set of predefined contingencies from a PowerWorld® Auxiliary Contingency file. Thus once you have defined a set of [contingencies](#) in Simulator and saved the list to a PowerWorld® Auxiliary file, you can specify the filename with this function and the [Simulator Automation Server](#) will load the contingencies into memory.

LoadContingencies(fileName, EString, {tAppend})

Parameter Definitions

Filename : String The name of the PowerWorld® Auxiliary file containing the defined contingencies. If the file cannot be found, an error will be returned through EString.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

tAppend : Boolean This parameter is optional. If you have more than one list of contingencies in separate files, you can append the lists together in the Simulator Automation Client by passing the Boolean value True in this parameter. The default if not passed is False, meaning any existing contingencies in the Simulator Automation Client would be cleared prior to loading the new list.

Example

LoadContingencies("c:\my files\ctgfile.aux", EString, True)

This function call will load a previously created list of contingencies that was saved in the PowerWorld Auxiliary file format. The file name string identifies the location and name of the auxiliary file. By passing the value True in the optional parameter tAppend, Simulator will read the list of contingencies and add them to any existing contingencies already loaded in Simulator. Contingencies in the list existing list with the same name as contingencies in the appending list will be replaced by the records in the appending list.

OpenCase Function (version 9)

The OpenCase function will load a PowerWorld® Simulator load flow file into the [Simulator Automation Server](#). If an error occurs trying to open the case, an error message will be returned through EString.

OpenCase(filename, EString)

Parameter Definitions

FileName : String The name of the PowerWorld® Simulator case file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name. If the file cannot be found or an error occurs while reading the file, an error message will be returned through EString.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

ProcessAuxFile Function (version 9)

The ProcessAuxFile function will load a PowerWorld® Auxiliary file into the [Simulator Automation Server](#). This allows you to create a text file (conforming to the PowerWorld® Auxiliary file format) that can list a set of data changes and other information for making batch changes in Simulator. If an error occurs while processing the auxiliary file, an error message is returned through EString.

ProcessAuxFile(filename, EString)

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

RunScriptCommand Function (version 9)

The RunScriptCommand function is used to execute a list of [script statements](#). The [script actions](#) are those included in the script sections of the [Auxiliary Files](#). If an error occurs trying to run a script command, an error will be returned through EString.

RunScriptCommand(Statements, EString)**Parameter Definitions**

Statements : String The block of script actions to be executed. Each script statement must end in a semicolon. The block of script actions should **not** be enclosed in curly braces.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

Example**RunScriptCommand("Entermode(PowerFlow); SolvePowerFlow;", EString)**

This function call will switch to the PowerFlow submode and then will solve the power flow of the present case.

SaveCase Function (version 9)

The SaveCase function is used to save a case previously loaded in the [Simulator Automation Server](#) using the [OpenCase](#) function. The function allows you to specify a file name and a format for the save file. If an error occurs while trying to save a case, an error message is returned through EString.

SaveCase(fileName, EString, {fileType}, {Overwrite})

Parameter Definitions

fileName : String The name of the file you wish to save as, including file path. No default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

fileType : String This parameter is optional. If you desire to save the case in a format other than the current PowerWorld Binary (pwb) format, you must specify the type of format in this parameter as a string. If the parameter is not passed, the format type defaults to the most recent version of PowerWorld binary (pwb) file. If a string is passed for this parameter, it must at least be the "PWB" string. An empty string will return an error. The following list is the currently supported list of string identifiers and the file types they represent.

"PTI23" PTI version 23 (raw)

"PTI24" PTI version 24 (raw)

"PTI25" PTI version 25 (raw)

"PTI26" PTI version 26 (raw)

"PTI27" PTI version 27 (raw)

"GE" GE PSLF (epc)

"IEEE" IEEE common format (cf)

"PWB70" PowerWorld Binary version 7.0 (pwb)

"PWB" PowerWorld Binary (most recent) (pwb)

Overwrite : Boolean This parameter is optional. If you do not want to overwrite a file with the same name as passed in filename, you must specify this parameter as False. Default is True.

SendToExcel Function (version 9)

The SendToExcel function can be called to send data from the [Simulator Automation Server](#) to an Excel spreadsheet. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few [field types](#) as desired that are supported by the type of object. The first time this function is called, a new instance of Excel will be started, and the data requested will be pasted to a new sheet. For each subsequent call of this function, the requested data will be pasted to a new sheet within the same workbook, until the workbook is closed. If an error occurs while trying to send data to Excel, an error message is returned through EString.

SendToExcel(tObjectType, filterName, EString, {tFieldList})

Parameter Definitions

tObjectType : String A string describing the type of object for which you are requesting data. No Default.

filterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used. Default is an empty string.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

TFieldList : Variant This parameter is optional. A variant array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If no array is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Example

SendToExcel("gen", "", EString, [pwBusNum, pwGenID, pwGenAGCable])

This function call will send the values of the fields in tFieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found. By specifying the fields in the optional parameter tFieldList, only the three field values for each generator will be returned. If the optional parameter tFieldList had been omitted, Simulator would have returned internally defined default information for the generators.

WriteAuxFile Function (version 9)

The WriteAuxFile function can be used to write data from the case in the [Simulator Automation Server](#) to a PowerWorld® Auxiliary file. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few field types as desired that are supported by the type of object. In addition, you can specify a new file name for each call to WriteAuxFile, or you can specify the same file name and append the data to the file. If an error occurs while trying to write the auxiliary file, an error message is returned through EString.

WriteAuxFile(fileName, filterName, tObjectType, EString, {tAppend}, {tFieldList})

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file you wish to save. No default.

filterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used. Default is an empty string.

tObjectType : String A string describing the type of object for which you are requesting data. No Default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

TAppend : Boolean This parameter is optional. If you have given a file name of an auxiliary file that already exists, then the file will either be appended to or overwritten according to the setting of this parameter. If the parameter is not passed, the Simulator Automation Server assumes false.

TFieldList : Variant This parameter is optional. A variant array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If no array is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Example

WriteAuxFile("c:\my files\myauxfile.aux", "", "gen", EString, False, [pwBusNum, pwGenID, pwGenAGCable])

This function call will send the values of the fields in tFieldList to a PowerWorld Auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found. By specifying the fields in the optional parameter tFieldList, only the three field values for each generator will be returned. If the optional parameter tFieldList had been omitted, Simulator would have returned internally defined default information for the generators. Since the tFieldList optional parameter was included, the tAppend optional parameter also had to be included, even though its default value is already False.

PowerWorld Object Variables (Version 9)

The ability to access power system data for different objects through various [Simulator Automation Server functions](#) is based on variables defined in Simulator that can be referred to as Object Field Variables. Each object (i.e. bus, generator, etc.) can have numerous fields associated with it. Each of these fields, in turn, has a variable associated with the field to enable access to the field for the purpose of acquiring or changing data. For example, the [GetParameters](#) function has a parameter called ParamList, which is intended to store a list of Object Field Variables for a particular type of object. When the function is called, the Simulator Automation Server will return the values associated with each particular field variable for the type of object specified. These field variables allow for complete flexibility by the user in specifying as many or as few fields for a particular object when acquiring or changing data.

Examples of Field Variables

PWBusGenMW

PWBusNum

Simulator has literally hundreds of parameters spanning numerous types of device and option specifications. Rather than list all of the field variables and the value they represent in this help file, we have enabled Simulator to automatically generate a text file containing the field variables and a description of what value the variable represents. PowerWorld Corporation highly recommends that you examine this list. To generate this text file, run PowerWorld Simulator and access the **Help** menu. Choose the option **Export Object Fields...** Specify a file name and location for saving the file, and click Save. Simulator will save out the field variables, the type of variable (string, integer, etc.), and a description of the value the field variable represents, with key fields for different objects marked with an asterisk. The field variables will also be split into sections based on the type of object they are valid for. Note that the same field variable may be available for more than one object, but that the value represented by the field variable might vary for different objects.

Examples of Field Variables in Listing

PwBusGenMW Real 'Bus Gen MW'

* PwBusNum Integer 'Bus Number' (* denotes key field in text file)

ATC_MaxLimElements Integer 'Max # Limiting Elements'

The last note on the Object Field Variables is that some of the object field variables are reused for more than one value for an object. For example, a transmission line has "from" and "to" buses associated with the line. Rather than have separate field variables for values at each terminal bus, the same field variable is used for both, with a colon followed by a number appended to the variable to signify which bus the value represents. As an example, consider the field variable for bus per unit voltage, which is PWBusPUVolt. Since there are two buses per line, the "From" bus voltage would be represented as PWBusPUVolt:0, and the "To" bus voltage would be represented as PWBusPUVolt:1. The enumeration of the field variables always starts with 0 for the first instance. You may note that in the text file of field values that you don't see any field variables with :0 appended to them. Since we did not want you to have to always append the :0 on all field variables, the default for a field variable with no appended :# is 0. Thus you would only need to be concerned with appending the :# for field variables that require a number greater than 0.

Topology Processing Overview

The Topology Processing tool is available only if you have purchased the Topology Processing add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering Topology Processing for Simulator.

Topology Processing is a Simulator add-on, which allows you to solve EMS-type, full-topology models in a numerically robust and transparent manner. Planning cases, traditionally used in the planning environment by tools such as Simulator, correspond to a less-detailed, electrically equivalent snapshot of the full-topology model.

Topology Processing extends Simulator applications traditionally used by planners so they can operate in an operations real-time environment.

Topology Processing is fully integrated with the functions of the Simulator base package including power flow, contingency analysis, and sensitivity calculations, as well as with the [OPF](#), [SCOPF](#), Transient Stability, [ATC](#), and [PVQV](#) add-ons.

Simulator's Topology Processing is different from traditional EMS topology processing, which requires converting the [full-topology model](#) into a consolidated, planning case, to numerically solve network applications such as power flow and state estimation. Simulator's Topology Processing dynamically obtains an equivalent [consolidated representation](#) of the full-topology model by dynamically reassigning the connection pointers of all of the devices. In Simulator, no "case conversion" takes place. The consolidated representation has the same topology and is electrically equivalent to the corresponding planning case.

Only the full-topology model is stored in memory. We call this novel implementation "Single-Model" Topology Processing.

Simulator's Topology Processing allows complete unification of power system planning and operations unification regarding network models and application environment. Typical uses of Topology Processing are:

- In the real-time production environment through [SimAuto](#) automation, yielding implementations of true real-time applications: real-time power flow, real-time contingency analysis (Security Assessment - SA), real-time transient stability (Dynamic Security Assessment - DSA), PVQV (Voltage Stability Assessment - VSA), real-time SCOPF (Security-Constrained Economic Dispatch - SCED).
- In near real-time operations by allowing simulation of real-time full-topology cases from the EMS.
- In the planning environment by allowing extensive study of the real-time system.

Simulator's Topology Processing consists of two main features:

- An algorithm that takes place behind the scenes, and that is used internally during numeric solutions such as power flow, ATC, etc.
- A tool to convert the real-time model to a traditional planning case that can be consumed by planning software not able to operate on full-topology models.

We invite you to continue reading the following Topology Processing topics:

[Full-Topology Model](#)

[Superbuses](#)

[Subnets](#)

[Consolidation](#)

[Primary Bus Priority Scheme](#)
[Illustration of Consolidation Process](#)
[Consolidated Representation](#)
[Consolidated Superbus View](#)
[Topology Processing Dialog](#)
[Power Flow Using Topology Processing](#)
[Ill-Conditioned Jacobian](#)
[Contingency Analysis Using Topology Processing](#)
[Incremental Topology Processing](#)
[Topology Processing in Applications](#)
[Saving the Planning Case](#)
[Planning Cases](#)
[Primary Bus Mapping](#)
[Breaker-to-Device Contingency Conversion](#)

Topology Processing: Full-Topology Model

Real-time applications require modeling the power network at a level of detail that includes all the switching devices such as circuit breakers and disconnects. For instance, an operator must have information about specific breaker statuses to coordinate device maintenance procedures.

These detailed models are referred to as “node-breaker” or “full-topology” models. This type of model cannot be solved directly in conventional planning applications because of the large number of very low impedance branches resulting from the switching devices. The power flow would be forced to use an [Ill-Conditioned Jacobian](#) matrix if these low-impedances were directly modeled. In order to solve the power flow on a full-topology model, Simulator dynamically obtains a [Consolidated Representation](#) of the power network by dynamically moving the device connection pointers.

Simulator models all type of branch objects as transmission line records. To aid in the consolidation, additional input parameters are necessary to properly identify the type of branch object that is being modeled and determine whether or not that branch can be eliminated during consolidation. These fields are accessible from the Topology group in the Available Fields for the [Line and Transformer Display](#) in the [Model Explorer](#). Two relevant input fields needed for Topology Processing are:

Branch Device Type

Type of branch specified as *Transformer*, *Line*, *Breaker*, *Disconnect* or *ZBR* (Zero-Impedance Branch).

Allow Consolidation of Branch

This is a *YES* or *NO* field, which tells Simulator to eliminate closed switching devices from the network model during Consolidation. Only branches that have a **Branch Device Type** equal to *Breaker*, *Disconnect*, or *ZBR* can be consolidated. If an element is of this type and is set not to be consolidated, it will be preserved in the model and its default impedance value will be used in the power flow equations.

Simulator models all of the system connection points: Busbars, Junctions, Terminals, etc. as buses. Topology Processing provides an optional input field for buses to aid in the consolidation:

Node Priority

Priority level of the bus to become a primary bus. This overrides the default Primary Bus Priority Scheme level for this bus.

Topology Processing determines two bus fields as part of the consolidation:

Primary Node

Bus to which all the device connection pointers in a Superbus will point to during Consolidation.

Node Neighbor List

If this is a Primary Bus, this contains a list of all buses that belong to the superbus in which this bus is contained. This entry is blank for buses that are not primary buses.

These three bus fields are available under the Topology group in the Available Fields for the [Bus Display](#) found on the [Model Explorer](#).

Topology Processing: Superbuses

A SuperBus in the [Full-Topology Model](#) is a group of buses connected through CLOSED switching. All the buses in a Superbus have the same voltage phasor, i.e., they correspond to the same electric point. As the status of switching devices change, superbuses change correspondingly, sometimes resulting in superbus merging or splitting.

Superbuses are set automatically when the case is read from the binary or aux file and they are updated at the beginning of a power flow solution or sensitivity calculation to capture possible changes in the status of switching devices. The details of the Superbus records is available in the [Model Explorer](#) under **Solution Details > Superbuses**.

A superbus in the [Full-Topology Model](#) would correspond to a bus in the electrically equivalent bus/branch planning case.

Each Superbus in a system belongs to only one [Subnet](#).

Topology Processing: Subnets

A Subnet is a group of buses connected through switching devices regardless of their open or closed status. Subnets are bounded by transmission line and transformer terminals. Subnets on a system hence do not depend on the status of breakers, but only on the presence of devices. Subnets do not change unless equipment is added or removed from the physical model.

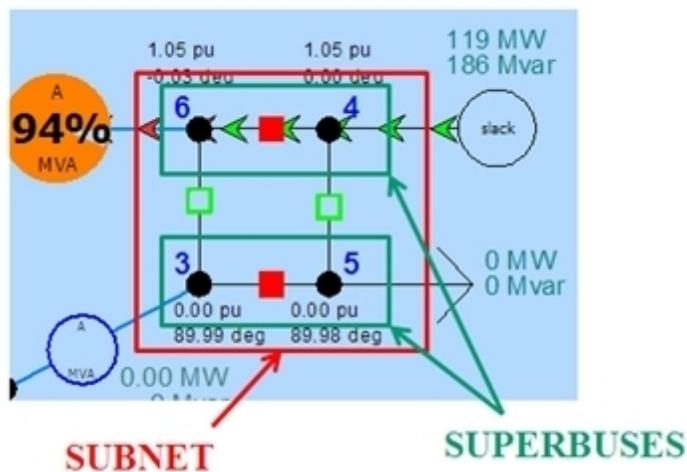
A Subnet can contain one or more [Superbuses](#).

A [Superbus](#) belongs to only one subnet.

Subnets and Superbuses are set automatically when the case is read from the binary or aux file.

The list of subnets and the details of subnet records can be seen on the [Model Explorer](#) under **Solution Details > Subnets**.

The example below shows a subnet that has four nodes. The shown breakers statuses results in two superbuses. If either breaker is closed, the two superbuses will merge.



Topology Processing: Consolidation

Consolidation consists in moving the connection pointers of all the devices within a Superbus to a specially selected bus called the Primary Bus, and hiding not needed switching devices and buses. The Primary Bus is selected based on a default [Primary Bus Priority Scheme](#).

Prior to consolidation, all the power system devices in a superbus, such as generators, loads, shunts and transmission line terminals are connected to their actual terminal buses in the physical system. After consolidation all the devices are connected to their corresponding Primary Bus. This makes the status and actually, the presence of switching devices in a superbus irrelevant, allowing the application to disregard them during the numerical solutions. We say that the full-topology model is in a consolidated state and we call the representation of the system after consolidation, the [Consolidated Representation](#).

After consolidation, the topology of the full-topology model is identical to that of an equivalent planning case and the models are electrically identical. However, in Simulator no case conversion takes place. Only the full-topology model exists in memory.

The following link contains an example illustrating the consolidation process: [Illustration of Process](#).

The user is never allowed to interact with the model while it is in a consolidated state.

The user does not have access to the Consolidated Representation unless it is saved as a planning case as described in the section [Saving the Planning Case](#).

Topology Processing Consolidation: Primary Bus Priority Scheme

During [Consolidation](#), the devices connected to the buses of a [superbus](#) change their connection pointers to point to the Primary Bus. In this manner, all the devices appear to be connected to primary buses only.

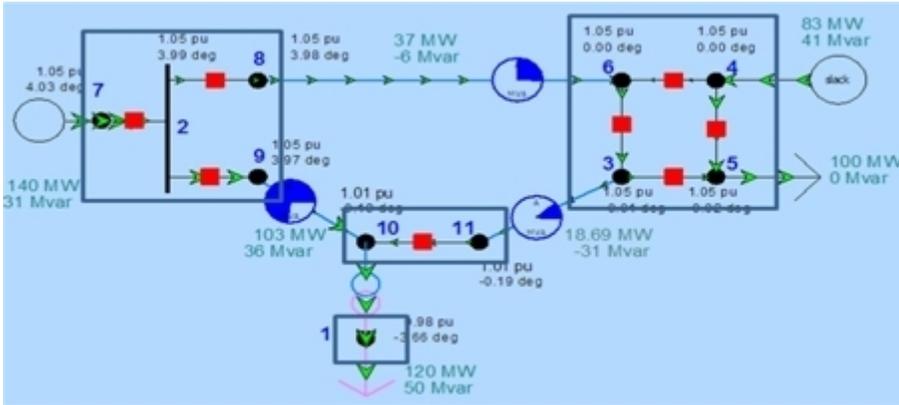
The selection of the primary bus takes place using the following priority scheme, in descending order of priority:

1. Slack bus (highest priority)
2. Multi-Terminal DC Line Terminals
3. Generator Regulated Bus
4. Switched Shunt Regulated Bus
5. LTC Regulated Bus
6. DC Line Terminal
7. Generator Terminal
8. Switched Shunt Terminal
9. Load Terminal
10. AC Transmission or Transformer Terminal
11. Breaker Terminals (lowest priority)

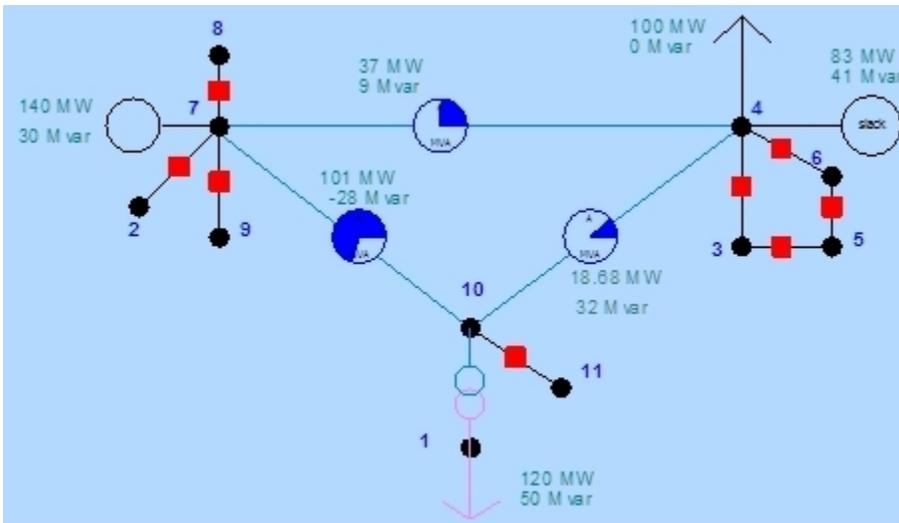
In case of ties, the minimum bus number is selected as primary bus.

Topology Processing Consolidation: Illustration of Process

Consider the following 11-bus, [full-topology model](#). Given the breaker statuses, this system would contain 4 [superbuses](#) shown in the next figure.



We apply [consolidation](#) to the system above considering that the primary buses of each superbuse are buses 1, 4, 7 and 10. The system topology at this point (which is never seem buy the user) would look as follows.



At this consolidated state, the breakers and the non primary bus buses could be removed without affecting the power flow solution.

Topology Processing: Consolidated Representation

A consolidated representation of a full-topology model is one obtained when the system is in a consolidated state. In this state all the connection pointers of devices and device terminals have been moved to the primary buses of the [Superbuses](#) as part of the [Consolidation](#) process.

The user does not have access to the Consolidated Representation records unless it is saved as a planning case as described in the section [Saving the Planning Case](#). However, the Consolidated Representation can be navigated using the [Consolidated Superbus View](#).

Topology Processing: Consolidated Superbus View

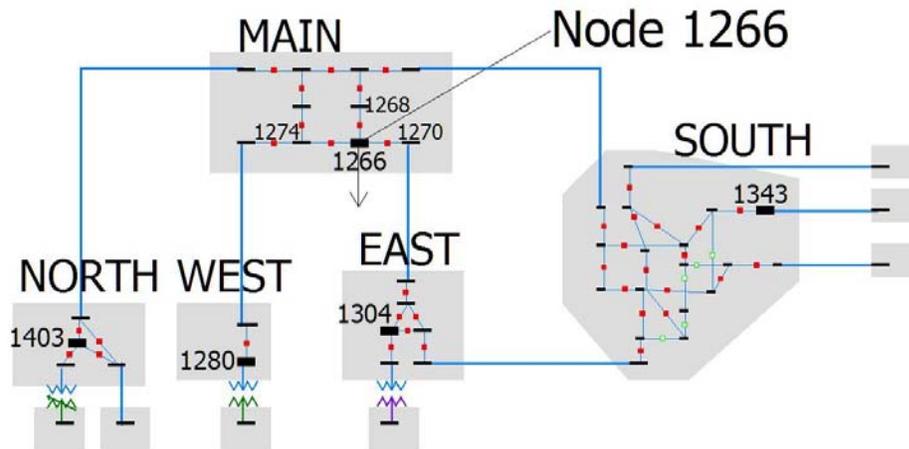
Simulator's Bus View provides a feature that allows a [Full-Topology Model](#) to be navigated using the [Consolidated Representation](#), in which only the primary buses are shown. This gives the appearance to the user of navigating the electrically equivalent planning case.

The Consolidated Superbus View is accessed from the [Bus View Display](#). If the current case is set up such that [superbuses](#) can be identified, there will be a drop-down box at the top of the Bus View Display that will allow switching between the Full Topology representation and the Consolidated Superbus representation.

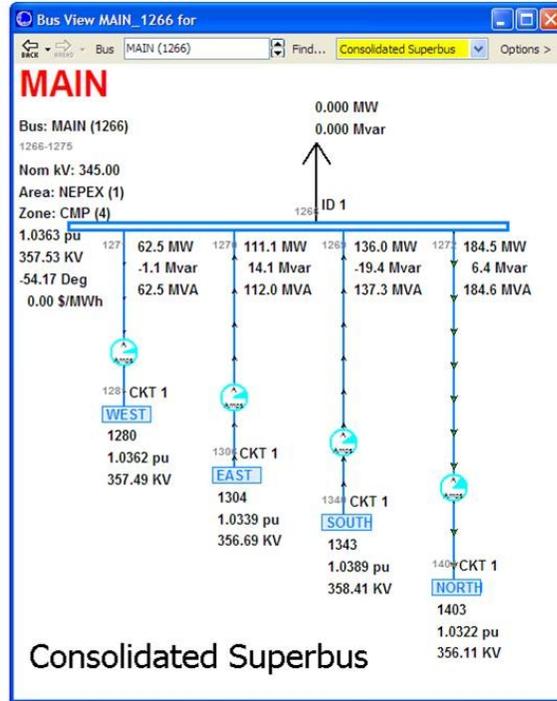
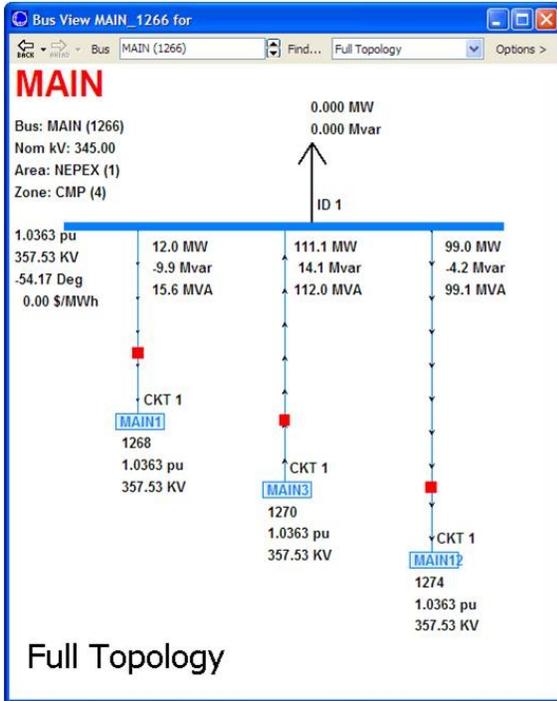
The Consolidated Superbus View has the following conventions:

- Shows radial-connected generator, load, and switched shunt buses above the main (primary) bus
- For series connections of buses, the main branch shown will be the first branch that is marked as NOT available for consolidation (i.e. AllowConsolidation = *NO*)
- All generator, load, and switched shunt devices will appear connected to their Primary Bus instead of the actual bus
- Only branches that connect different superbuses will be shown. This means that closed circuit breaker will not appear on the consolidated superbus view because they connect two buses that are in the same superbus.

An example full-topology model is shown below. Each gray region represents a superbus. The large buses and numbers in each superbus represent the primary bus for that superbus.

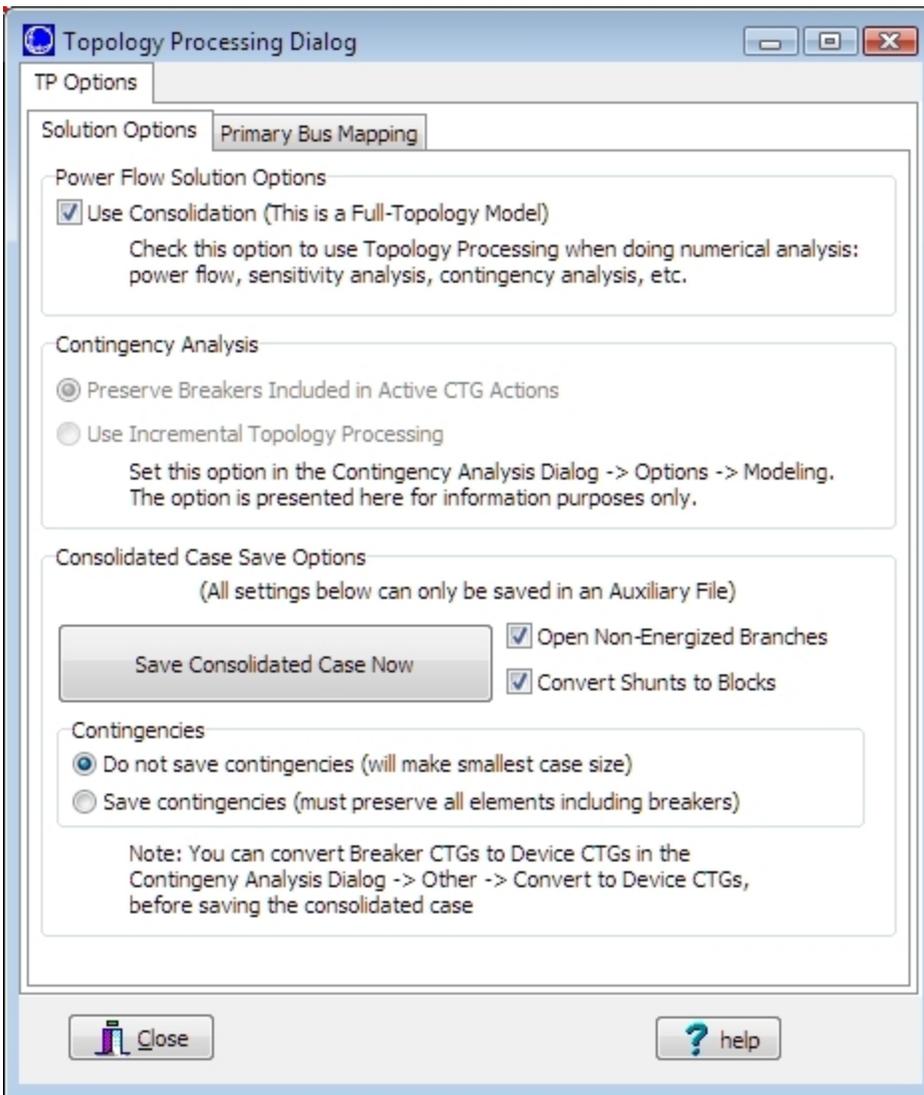


The associated full-topology bus view and the consolidated superbus view for the above model is shown below:



Topology Processing Dialog

To open the Topology Processing Dialog select **Topology Processing** found on the [Add Ons](#) ribbon tab in the **Topology Processing** ribbon group. This dialog includes options to solve the [Power Flow Using Topology Processing](#), perform [Contingency Analysis Using Topology Processing](#), and for [Saving the Consolidated Case](#).



Power Flow Solution Options

Use Consolidation (This is a Full-Topology Model)

This option is used for all Simulator [applications](#): power flow, contingency analysis, sensitivity calculations, etc. This option should ALWAYS be checked for real-time cases, while it should be unchecked for planning cases. If a full-topology model was read and this option is not checked, the power flow will try to solve the model without removing the switching devices resulting in a

very large [ill-conditioned Jacobian](#). This option is also found on the [Power Flow Solution Common Options](#) page of the [Simulator Options](#) dialog.

Contingency Analysis

These options are shown on this dialog for informational purposes only. These can be accessed and set from the [Contingency Analysis dialog](#) on the [Options](#) tab and [Modeling](#) sub-tab. These options are described in more detail in the [Contingency Analysis Using Topology Processing](#) topic.

Consolidated Case Save Options

These options allow the saving of the consolidated representation as a traditional planning case. More detail about these options is provided in the [Saving the Planning Case](#) topic.

Options found on the **Primary Bus Mapping** tab are described in the [Primary Bus Mapping](#) topic.

Topology Processing: Power Flow Using Topology Processing

In order to perform numerical solutions using Topology Processing, Simulator does the following:

[Consolidates](#) the system relocating the devices and removing not needed breakers and non primary buses.

Performs the numerical calculation.

De-consolidates the system by restoring breakers and relocating the devices to their original buses. This step also maps the values of the primary buses to all the buses in the [superbus](#).

This is transparent to the user, who only needs to tell Simulator to use consolidation. The option that needs to be set to use consolidation is found on the [Topology Processing Dialog](#) and also on the [Power Flow Solution](#) page of the [Simulator Options](#) dialog.

Topology Processing: Ill-Conditioned Jacobian

Matrix **A** in the linear equation $\mathbf{Ax} = \mathbf{b}$ is ill-conditioned if small changes in the entries of the vector **b** radically change the solution vector **x**. Ill-conditioned matrices are particularly troublesome in iterative solvers such as AC power flow algorithm implementations. One iteration of the AC power flow solves the linear equations $\mathbf{J} \Delta \mathbf{x} = \Delta \mathbf{y}$ where **J** is the Jacobian matrix, $\Delta \mathbf{x}$ is the vector of voltage magnitude and angle differences and $\Delta \mathbf{y}$ is the vector of real and reactive power mismatches.

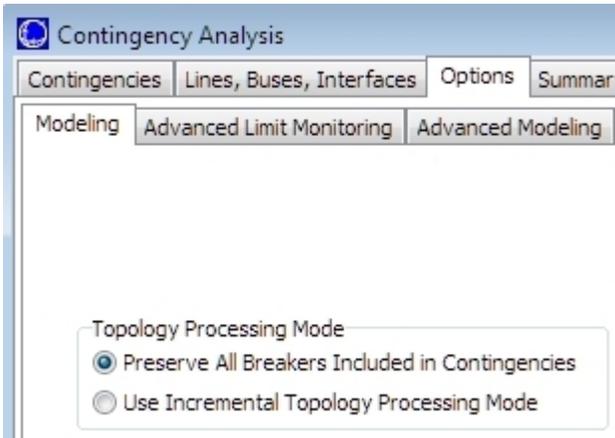
A power flow Jacobian can become ill-conditioned if the system branches present reactances that differ in orders of magnitude, such as very short lines or zero impedance branches. Because the reactance of switching devices is negligible compared to that of transmission lines and transformers, modeling but a short number of switching devices in the Jacobian is prohibitive. The lower the value of the reactance, the more ill-conditioned the Jacobian will become. On the other hand, if the assumed reactance value is not too low, the solution would be inaccurate resulting in larger bus voltage angles. In Simulator switching devices that are present in [Full-Topology Models](#) are handled by [Topology Processing](#), producing an exact and robust numerical solution.

Topology Processing: Contingency Analysis Using Topology Processing

Simulator Topology Processing allows solving contingencies in the [Full-Topology Model](#).

EMS systems model contingencies through a list of real system actions. For instance, opening a line involves opening the set of breakers that electrically isolate the device. The same contingency action in a planning environment would be modeled as a single contingency action: OPEN LINE.

The option to determine how contingencies should be modeled within topology processing is found on the [Contingency Analysis dialog](#) on the [Options](#) tab and [Modeling](#) sub-tab.



Preserve All Breakers Included in Contingencies

If this option is selected, contingency analysis will not treat contingency breakers as zero impedance branches and will keep all of them in the model. Hence, the consolidated representation is obtained only once upfront before the first contingency, and this representation is used to solve all the contingencies. Using this option, the consolidated representation is smaller than the full model, but is not the smallest possible. If the number of contingencies and breaker actions is large, then many breakers will be preserved in the case. If the percentage of preserved breakers is significant, this may result in an [Ill-Conditioned Jacobian](#) during the contingency solution.

Use Incremental Topology Processing Mode

In this option, contingency analysis uses an [Incremental Topology Processing](#) approach to solve each contingency. Since usually the number of contingency actions in a contingency is small (usually around 4 to 10), only a small number of [Subnets](#) are affected by a contingency. The topology of all the other subnets remains unchanged. When this option is used, Simulator does a full consolidation before the first contingency and then for each contingency, the topology of just the affected subnets is re-processed (determining superbuses and primary buses, and updating the connection pointers). Incremental Topology Processing provides a more robust solution and should be used whenever possible.

Incremental Topology Processing cannot be used for applications that use linearization and that require a constant size of the bus array, such as sensitivity analysis. For instance, ATC will automatically use the contingency analysis **Preserve Breakers Included in Contingencies** option.

Topology Processing: Incremental Topology Processing

Incremental Topology Processing is a technique used to update the [Consolidated Representation](#) of the system in a very fast manner. It is particularly useful during [Contingency Analysis](#), where it does the following:

Consolidate entire case

For each contingency do

Apply switching device actions and identify subnets with changed topology

Expand affected subnets to full-topology detail

Reconsolidate affected subnets with post-contingency topology

Solve post-contingency power flow

Determine and store limit violations

Reset affected subnets to pre-contingency topology

Restore pre-contingency power system state

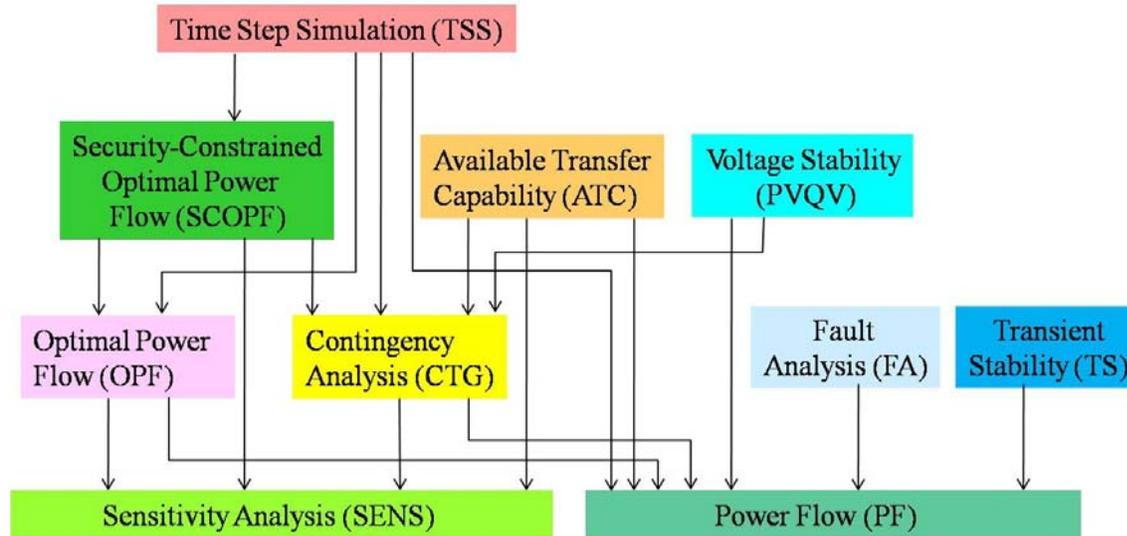
End

Deconsolidate entire case

Incremental Topology Processing is very fast, taking on the order of a few milliseconds.

Topology Processing: Topology Processing in Applications

Simulator applications such as ATC and OPF use a hierarchical scheme where [Consolidation](#) takes place only at the beginning of the numerical solution process. Consolidation in lower functions takes place only if it has not taken place already in higher-level functions.



Following completion of the application, de-consolidation occurs in the function level in which the consolidation originally occurred. This process is very similar to the one described for [solving the power flow using topology processing](#). There can be some exceptions to this when using Contingency Analysis. The [Contingency Analysis Using Topology Processing](#) topic provides more details on this.

Once the input data is set in the [Full-Topology Model](#) fields, the applications work in the same manner as with regular [Planning Cases](#).

Topology Processing: Saving the Planning Case

Topology Processing allows you to save the [Consolidated Representation](#) of the [Full-Topology Model](#) as a traditional [Planning Case](#) (bus/branch model), in any of the [formats](#) supported by Simulator.

Although the saved case can be reopened in Simulator as a regular case, the user should note that the topology of this case will depend on the statuses of the breakers in the Full-Topology Model. If the statuses of the switching devices in the Full-Topology Model are continuously changing as is the case when the case comes from a real-time Retriever implementation, then the topology of the saved cases will be slightly different minute to minute.

The saving tool is available on the [Topology Processing Dialog](#) in the Consolidated Case Save Options section.

Consolidated Case Save Options
(All settings below can only be saved in an Auxiliary File)

Save Consolidated Case Now

Open Non-Energized Branches

Convert Shunts to Blocks

Contingencies

Do not save contingencies (will make smallest case size)

Save contingencies (must preserve all elements including breakers)

Note: You can convert Breaker CTGs to Device CTGs in the Contingency Analysis Dialog -> Other -> Convert to Device CTGs, before saving the consolidated case

Save Consolidated Case Now

Click this button to save the case in any of the supported [formats](#).

Open Non-Energized Branches

If the terminal buses of a transmission line or transformer are not energized, then assume the element is open.

Convert Shunts to Blocks

Some systems have several shunts in a superbus, and some formats support only one shunt per bus. This option allows modeling those shunts as an aggregate device.

Do not save contingencies (will make smallest case size)

Use this option if contingency analysis will not take place on the saved case.

Save contingencies (must preserve all elements including breakers)

All the contingencies, the contingency actions, and all the switching devices included in contingency actions will be preserved and saved in the consolidated case. It makes a big difference to perform a [Breaker-to-Device Contingency Conversion](#) before saving the case. Usually most of the breaker contingencies can be converted to device contingencies requiring just a small number of breakers to be preserved in the case. Some planning tools have limitations regarding the number of low impedance branches that can be modeled and will not be able to read and solve the [Planning Case](#) if it contains a large number of switching devices.

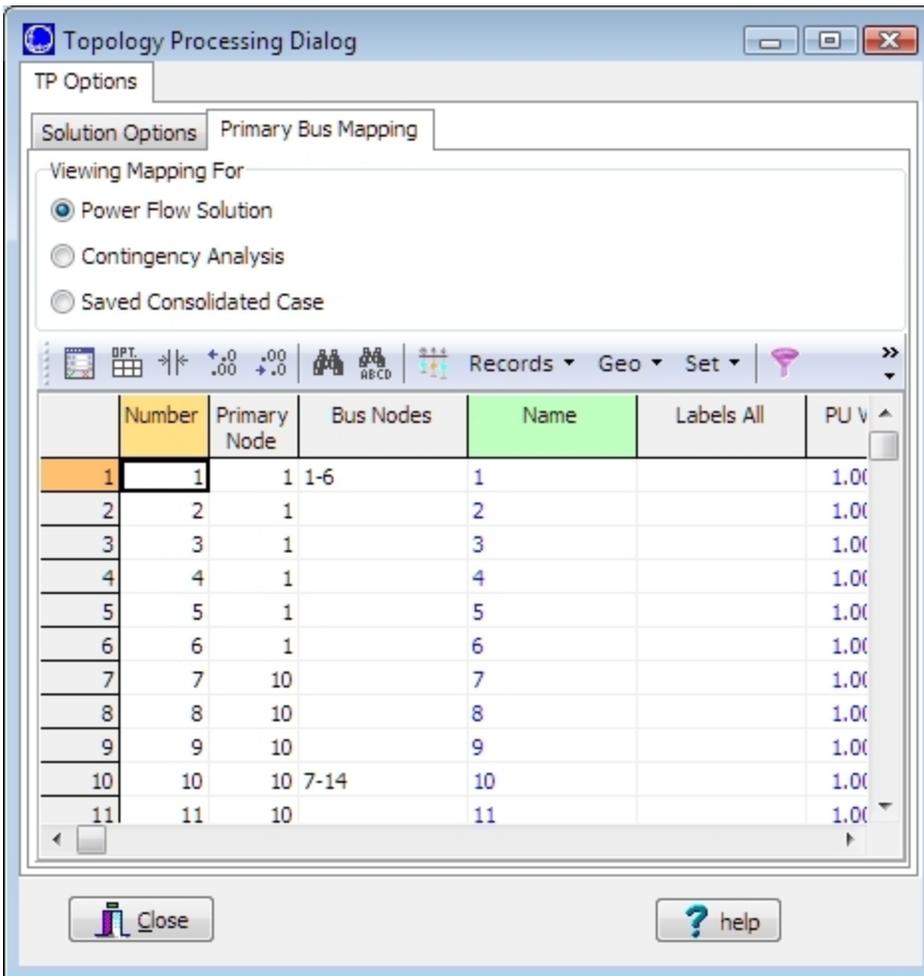
Topology Processing: Planning Cases

Planning Cases are a high-level, simplified view of the actual network, in which all the switching devices have been removed. Planning Cases use a bus/branch model that uses the bus numbers as key fields for all the system elements. [Full-Topology Models](#) on the other hand, are detailed representations of the physical system that include circuit breakers and disconnects. Simulator Topology Processing allows complete model unification and provides a unified application environment to simulate both types of models.

Topology Processing: Primary Bus Mapping

Sometimes, it is necessary to explore or export the topology relations between [superbuses](#) and buses in the [Full-Topology Model](#) and its [Consolidated Representation](#). Because different sets of switching devices can be preserved by Simulator when solving the [Power Flow Using Topology Processing](#), when solving the [Contingency Analysis Using Topology Processing](#), and when [Saving the Planning Case](#), there will be different mappings for each one of these actions.

Primary Bus Mapping options are available on the **Primary Bus Mapping** tab of the [Topology Processing Dialog](#).



This display shows a detailed mapping of superbuses to buses. By changing the selection for which mapping to view, the corresponding mapping that would take place is shown in this [case information display](#).

Power Flow Solution

Provides the mapping used when solving the power flow.

Contingency Analysis

Provides the mapping used when solving contingencies.

Saved Consolidated Case

Provides the corresponding topology relations between the full-topology model and the consolidated case to be saved.

This mapping can be saved as an auxiliary file from the local menu of the display (right-click and choose **Save As > Auxiliary File...**) or from the [case information toolbar](#).

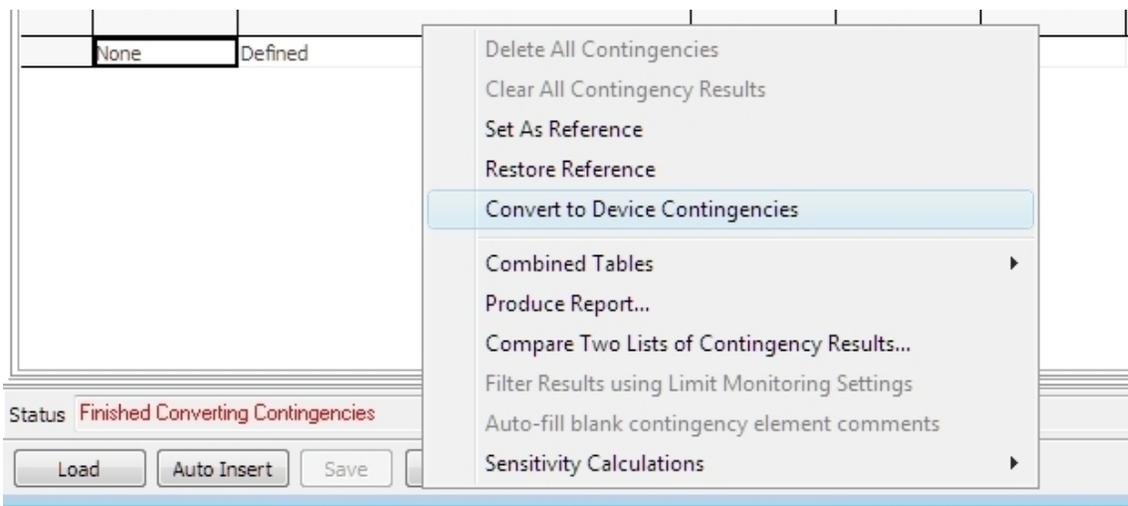
Topology Processing: Breaker-to-Device Contingency Conversion

While operation contingencies are defined using breaker actions, planning cases use simplified device [contingency actions](#). For instance, the outage of a single transmission line is modeled in EMS systems as the set of the actual circuit breaker actions that physically de-energize the transmission line. On the planning side, the same contingency is modeled as a single contingency action related to the transmission device: OPEN LINE.

A feature of PowerWorld's [Topology Processing](#) is the ability to convert operations-type, circuit breaker contingencies to their electrically equivalent device contingencies for single outage, multiple outages, or complex post-contingency topologies. While both the breaker contingencies and the device contingencies can be solved on the [Full-Topology Model](#), breaker contingencies MUST be converted to device contingencies in order to solve them on the consolidated [Planning Case](#).

Some complex breaker contingencies cannot be converted to electrically equivalent topologies by simply changing the energization status of devices. A subset of the breakers involved in those contingencies may need to be preserved in the case to entirely reproduce the electric topology.

The Breaker-to-Device Contingency Conversion is launched from the [Contingency Analysis Dialog](#) > **Other** button > **Convert to Device Contingencies**. This routine will convert all the contingencies in the list. The contingency analysis status box will indicate the progress of the contingency conversion process. The log will also provide messages during the conversion.



The new list of converted device contingencies will replace the original list of breaker contingencies previously displayed on the [Contingency Analysis Dialog](#). The list of device contingencies can be solved directly on the [Full-Topology Model](#). On average, device contingencies include less device actions per contingency compared to breaker actions. For instance, a single line outage usually requires opening several circuit breakers, whereas the device contingency equivalent requires a single OPEN LINE contingency action.

In order to reflect the conversion process, the actions in the [Contingency Definitions](#) include device actions with the name of the device: *Line*, *Transformer*, *Load*, *Gen*, etc. and also *Breaker* if breakers need to be preserved for a given contingency.

Transient Stability Overview

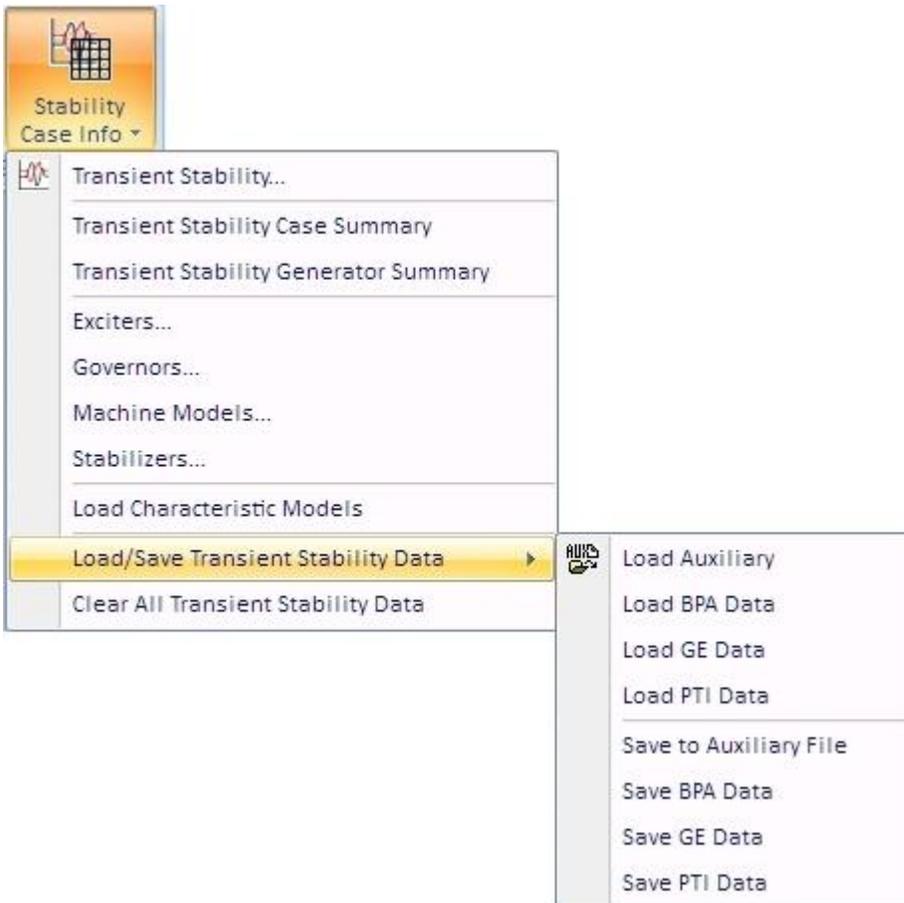
The Transient Stability tool is available as an add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for more details.

Transient stability studies analyze the system response to disturbances such as the loss of generation, line-switching operations, faults, and sudden load changes in the first few seconds following the disturbance. Following a disturbance, synchronous machine frequencies undergo transient deviations from synchronous frequency. The objective of a transient stability study is to determine whether or not machines will return to synchronous frequency following a disturbance.

The [Transient Stability Analysis dialog](#) provides access to all options and results and controls the analysis. The [Stability Case Info menu](#) provides access to stability models and allows the import and saving of models. Models can also be defined for individual generators and loads on their information dialogs or through the **Transient Stability** folder in the [Model Explorer](#).

Transient Stability Case Info Menu

The Stability Case Info menu is available by selecting **Stability Case Info** in the **Transient Stability (TS)** ribbon group on the [Add Ons](#) ribbon tab.



The menu provides direct access to several case information displays that show default columns related to input and output of the transient stability tool. There are also options available for loading and saving transient stability models.

Transient Stability

Choose this option to open the [Transient Stability Analysis dialog](#).

Transient Stability Case Summary

Choose this option to view a summary of transient stability models defined in the case. The summary includes the model name, whether the model is active or inactive, and whether or not the model is fully supported in Simulator.

Transient Stability Generator Summary

Choose this option to view a summary of all generators in the case and any stability models that have been defined for each generator.

Exciters, Governors, Machine Models, Stabilizers, Load Characteristic Models

Choosing any of these options will open a case information display for the appropriate model type. This same information can be accessed through the **Transient Stability** folder on the [Model Explorer](#).

The stability case information displays can be used to change and define new models with the case.

Load/Save Transient Stability Data

Choose the appropriate option to save or load transient stability model data in the desired format. Models are supported in the Simulator auxiliary file format (*.aux), BPA dynamics data format (*.swi), GE dynamics data format (*.dyd), and PTI dynamics data format (*.dyr).

Clear All Transient Stability Data

Choose this option to delete all transient stability models.

Transient Stability Analysis Dialog

To display this dialog, go to the [Add Ons](#) ribbon tab and select **Transient Stability** from the **Transient Stability (TS)** ribbon group.

The dialog is broken down into several pages designed to take the user through the setup, processing, and analysis of results. The pages are as follows:

[Simulation Control](#)

[Options](#)

[Results to Save](#)

[Plots](#)

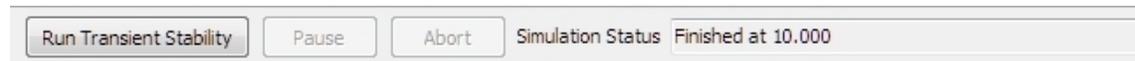
[Results](#)

[States/Manual Control](#)

[Validation](#)

SMIB Eigenvalues (not available yet)

There are several buttons at the top of the dialog that are available regardless of the selected page.



Run Transient Stability

Click this button to start the analysis once all options and models have been set.

Pause

Click this button to pause a study that is in progress.

Abort

Click this button to stop a study that is in progress.

Simulation Status

Indicates the progress of a study.

There are several buttons at the bottom of the dialog that are available regardless of the selected page.

Save Auxiliary...

Clicking this button will prompt for a filename in which to save an [auxiliary file](#). Use this option to store any results and settings that need to be retained for future use or use with different power flow cases. Transient stability-specific results and option settings can optionally be saved with the PowerWorld binary file (*.pwb).

Load Auxiliary...

Clicking this button will open a dialog to allow the user to select an [auxiliary file](#) to load. This is intended to be used for loading any relevant option settings to be used during the transient stability analysis. The dialog will be updated according to the option settings loaded from the file.

Transient Stability Analysis: Simulation Control

The Simulation Control page is available on the [Transient Stability Analysis dialog](#).

Simulation Control

Simulation Time Values

Start Time (seconds) 0.000 Specify Time Step in
 Seconds
 Cycles

End Time (seconds) 10.000
 Time Step (seconds) 0.020000

Transient Stability Events List
 The below list shows the transient stability faults and other events.

	Time (Seconds)	Enabled	Object	Event Description
1	1.00000	YES	Bus '5'	FAULT 3PB SOLID
2	1.10000	YES	Bus '5'	CLEARFAULT

Simulation Time Values

Start Time, End Time

Specify the start and end time of the analysis in seconds.

Time Step

The time step is entered in either seconds or cycles (60 cycles/second) depending on the setting of the **Specify Time Step in** option. This value will change to reflect the appropriate units if the **Specify Time Step in** option is changed.

Specify Time Step in

The **Time Step** can either be specified in seconds or cycles (60 cycles/second). Changing this option will update the value specified in **Time Step** to reflect the appropriate units.

Transient Stability Events List

Insert Event

Click this button to open the [Transient Stability Event Dialog](#).

Clear All Events

Click this button to delete all currently defined events.

Event Table

This table provides a listing of all transient stability events that are currently defined. This is a [case information display](#) and provides the case information toolbar and local right-click menu for options. New events can be inserted through the **Insert** menu/toolbar item. Events can be

deleted through the **Delete** menu/toolbar item. Use the **Show Dialog** option to open the [Transient Stability Event Dialog](#) to modify the event.

Transient Stability Analysis: Options

The Options page is found on the [Transient Stability Analysis dialog](#). The Options page is broken down into several sub-tabs:

[General](#)

[Power System Model](#)

Transient Stability Analysis Options: General

The General sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#).

Storage in PWB File

The user can select to just store the transient stability parameters, store the parameters results, or not store either the transient stability parameters or results in the *.pwb file. Storing transient stability information in the *.pwb file makes it convenient to open a case and run the transient stability analysis, but storing this information, especially the results, could significantly increase the size of the *.pwb file. Auxiliary files can also be used for storing the parameters and results instead of, or in addition to, storing this information in the *.pwb file.

MVA Base for Input/Display of Generator Values

Specify the MVA base to use for generators: either the MVA base value entered for each individual generator or the MVA base value specified for the case.

Identify Buses in Events and Results by

Choose how to identify any buses that appear in transient stability events and results. Buses can be identified by number or name or a combination of both. The nominal kV of the bus can also be used as part of the identifier.

Automatic Update and Transfer Results to Power Flow Options

Specify the interval in time steps to update the displays or when to transfer results to the power flow model. Also, specify whether or not the displays should be updated and when/if results should be transferred to the power flow model.

When Case has a Transient Stability Model Show Confirmation Dialog

If in the middle or at the end of a transient stability simulation the user tries to save the case, Simulator can prompt whether to save the case with the power system state after the transient stability simulation or to save the case with the power system state in place prior to starting the

transient stability analysis. This option determines when the prompt will appear. If not prompted, the state will remain in the state following the transient stability simulation.

Validation

Determine how to proceed if an unsupported transient stability model is defined for any element. An error will prevent the transient stability analysis from running and provide a message. A warning will set the status of any unsupported models to inactive and provide a message. No error or warning will set the status of any unsupported models to inactive but provide no message.

Any messages that result from the validation will show up on the [Validation](#) page.

Saving Min/Max Values

Minimum and maximum values for several bus and generator parameters will be stored during the analysis. This option determines when the recording of these minimum and maximum values should begin. If choosing to use a Custom Time, a field will appear that will allow entry of a specific time in seconds.

Save in PWX Format

Load Results from PWZ Format

Transient Stability Analysis Options: Power System Model

The Power System Model sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#).

The screenshot shows the 'Power System Model' sub-tab of the 'Options' dialog. It contains the following sections and controls:

- Power System Values:** Nominal System Frequency (Hz) set to 60.0; System MVA Base set to 100.00.
- Infinite Bus(es) Modeling:** Radio buttons for 'No infinite buses' (selected) and 'Model the power flow slack bus(es) as infinite buses'.
- Angle Reference (Only Used If There Are No Infinite Buses):** Radio buttons for 'Average of Generator Angles' (selected), 'Weighted Average of Generator Angles', 'Specified Angle Reference Generator Terminal Angle', 'Specified Angle Reference Generator Internal Angle', and 'Synchronous Reference Frame (No Angle Shift)'. Below are fields for 'Maximum Allowable Angle Difference (degrees)' (1080.0), a checkbox for 'Initialize with Reference Angle at Zero', and a dropdown for 'Angle Reference Generator' (1 (Bus1) #1).
- Power Flow Solution:** Fields for 'Solution Tolerance (MVA)' (0.100) and 'Maximum Iterations' (25).
- Governor Fast Valving:** Radio buttons for 'Global Fast Valving Option' (selected), 'Initiate on Freq Deviation > (rad/sec)', 'Initiate on Fault after Delay of Seconds', and 'Do Not Initiate'. A field for 'Fast Valving Parameter (rad/sec or sec)' is set to 0.1.
- Loading Modeling:** Radio buttons for 'Default Load Model' (selected), 'Constant Impedance', 'ZIP Model from Power Flow Model', 'Constant Current', and 'Constant Current P, Impedance Q'.
- Minimum Per Unit Voltages for:** Fields for 'Constant Power Models' (0.700) and 'Constant Current Models' (0.500), with a 'Change' button.

Power System Values

Nominal System Frequency (Hz)

Set the nominal system frequency in hertz. The default is 60 Hz.

System MVA Base

MVA base for the system. The value cannot be changed here and is shown for informational purposes only. To change this value, go to the [Power Flow Solution General page](#) found under [Simulator Options](#).

Infinite Bus(es) Modeling

Select whether or not to use infinite buses. If using infinite buses, the power system slack bus(es) will be used as the infinite buses. If not using infinite buses, then an **Angle Reference** must be selected.

At infinite buses the angle does not change. When using infinite buses, these are used as the angle reference.

When storing angles during the analysis, the angle stored is the difference between the actual angle at the bus and the angle reference. Results can also be stored for the angle at a bus without the shift due to the angle reference. These fields are labeled with **No Shift** as part of the field name.

Angle Reference (Only Used If There Are No Infinite Buses)

Select the angle reference to use if the option for **Infinite Bus(es) Modeling** is set to *No infinite buses*. If choosing one of the options for specifying a generator, the **Angle Reference Generator** option must be set.

When storing angles during the analysis, the angle stored is the difference between the actual angle at the bus and the angle reference. Results can also be stored for the angle at a bus without the shift due to the angle reference. These fields are labeled with **No Shift** as part of the field name.

Maximum Allowable Angle Difference (degrees)

This parameter determines the maximum allowable angle difference between any two generator rotor angles in the case. If this value is exceeded, the transient stability analysis will stop.

Initialize with Reference Angle at Zero

This initializes the reference angle, regardless of the method selected for determining the reference angle, to zero at the beginning of the analysis. In practice this is done by recording the starting reference angle. At each time step the reference angle is determined and then this is offset by subtracting the original reference angle value from the current reference angle value and this new value becomes what is used as the reference angle at each time step.

Angle Reference Generator

This option must be set if the option for **Angle Reference** is set to one of the choices for using a specific generator.

Power Flow Solution

Solution Tolerance (MVA)

Specify the convergence tolerance used in the transient stability's power flow algorithm. This option setting only affects the transient stability power flow algorithm and does not affect the power flow model.

Maximum Iterations

Specify the maximum number of iterations allowed in the transient stability's power flow algorithm. This option setting only affects the transient stability power flow algorithm and does not affect the power flow model.

Governor Fast Valving

Fast Valving Parameter (rad/sec or sec)

Default Load Model

Specify what load model to use by default in the transient stability simulation when the load does not have a load model characteristic defined.

Minimum Per Unit Voltages for

When modeling loads as **Constant Power Models** or **Constant Current Models**, the load will start to fall off by a particular function if the voltage falls below the specified value. The values specified here are the same as the values for minimum voltages specified on the [Power Flow Solution Advanced Options page](#) found under [Simulator Options](#). Clicking the **Change** button will open the Power Flow Solution Advanced Options page.

Transient Stability Analysis: Results to Save

The Results to Save page is found on the [Transient Stability Analysis dialog](#). Settings on this page determine which results will be stored when the transient stability analysis is run. Plots of the results can also be defined from this page.

From Selection:	Number of Bus	Name of Bus	ID	Area Name of Gen	Save All	Rotor Angle	Rotor Angle, No Shift	Speed	Mech Input	MW Terminal
1	1	Bus 1	1	1	YES	NO	NO	NO	NO	NO
2	2	Bus 2	1	1	YES	NO	NO	NO	NO	NO
3	3	Bus 3	1	1	YES	NO	NO	NO	NO	NO

Data to Save

These options are currently a work in progress. What results are actually saved are dictated by the settings in the **Tables of Devices**. The Data to Save options currently make setting some of these options for the tables easier.

Currently, choosing the **Save All Results** option will set the **Save All** field to *YES* for all generators and buses.

Save Results Every n Timesteps

Set this option to specify at which timesteps results will be stored.

Save Results for Open Devices

Check this box to save results for devices that are currently open. It is possible that a device that is currently open could be closed during the transient stability analysis so this device should not be excluded from the results.

Tables of Devices

Results for Generators, Buses, Loads, Branches, DC Lines, Areas, and Zones can be stored during the transient stability analysis. Each type of object has a list of transient stability related parameters that can be stored. Toggling the appropriate field to *YES* for an object will store that parameter for the selected object. Each object also has a **Save All** field. If this field is set to *YES* for a particular object, all transient stability related fields will be stored for that particular object without having to set all of the fields to *YES* for all of the parameters.

All results are presently stored in memory. This means that a memory limitation could be reached is storing a large number of results.

Plotting

Transient stability results are usually interpreted with the assistance of some sort of plot. There can be a large amount of data to sort through and options on this page as well as the [Plots](#) page are designed to assist in setting up meaningful plots. Plot Definitions must be created in order to view a plot. The plot buttons contained on the Results to Save page assist in creating Plot Definitions outside of the Plots page. See the [Plots](#) page for more detailed information about Plot Definitions and plot parameters. Once options on the Results to Save page are used to create plots, the plot definitions will show up on the [Plots](#) page.

In order to make a plot, objects and fields to include in the plot must be specified. How many objects and fields are selected will determine which of the following buttons are enabled and how the plot will be designed. To select objects and fields for a plot, simply select cells in the appropriate device table that include the fields and objects to show on a plot.

For example, to plot all generator rotor angles on a single plot, go to the Generators tab and select all of the records under the Rotor Angle field. After selecting the objects and fields, click the appropriate plot button, as described below, to create the plot definition.

From Selection:	Number of Bus	Name of Bus	ID	Area Name of Gen	Save All	Rotor Angle	Speed	Rotor Slip
	1	1 Bus 1	1	1	YES		NO	NI
	2	2 Bus 2	1	1	YES		NO	NI
	3	3 Bus 3	1	1	YES		NO	NI

As another example, to plot only the rotor angles and terminal MW value of only generators 1 and 2, go to the Generators tab. Move the MW Terminal field so that it is next to the Rotor Angle field. This can be done by using the [Display/Column Options dialog](#) or holding down the CTRL key and dragging a column to move it to a different place in the table. Select the records for generator 1 and 2 under both the Rotor Angle and MW Terminal fields. After selecting the objects and fields, click the appropriate plot button, as described below, to create the plot definition.

The screenshot shows a software interface with a menu bar (Generators, Buses, Loads, Branches, DC Lines, Areas, Zones) and a toolbar with various icons. Below the toolbar is a table with the following data:

From Selection:	Number of Bus	Name of Bus	ID	Area Name of Gen	Save All	Speed	Rotor Angle	MW Terminal	Mec
	1	1 Bus1	1	1	YES	NO			NO
	2	2 Bus 2	1	1	YES	NO			NO
	3	3 Bus 3	1	1	YES	NO	NO	NO	NO

Below the table are three buttons: 'Make Plot', 'Make Plot Group by Field', and 'Make Plot Group by Object'.

Make Plot

This button is enabled when at least one object record for a transient stability field is selected. This option will create a plot definition with a single subplot with all selected objects and fields appearing on the same axis group.

Make Plot Group by Field

This button is enabled when more than one transient stability field is selected. This option will create a plot definition with a single subplot with a separate axis group for each of the selected fields. All selected objects will appear on a given axis group.

Make Plot Group by Object

This button is enabled when more than one transient stability object is selected. This option will create a plot definition with a single subplot with a separate axis group for each of the selected objects. All selected fields will appear on a given axis group.

Transient Stability Analysis: Plots

The Plots page is found on the [Transient Stability Analysis dialog](#). Settings on this page determine which results will be plotted and when the plots will be made visible during the transient stability analysis.

A few key terms that define the plot components need to be defined before proceeding with the plot definition explanation:

Key Terms

Plot

A single window showing plotted results. A single plot can contain any number of subplots.

Subplots

This represents the actual chart. This contains the horizontal axis as well as one or more axis groups representing the vertical axis. For many plots, there will only be one subplot.

Axis group

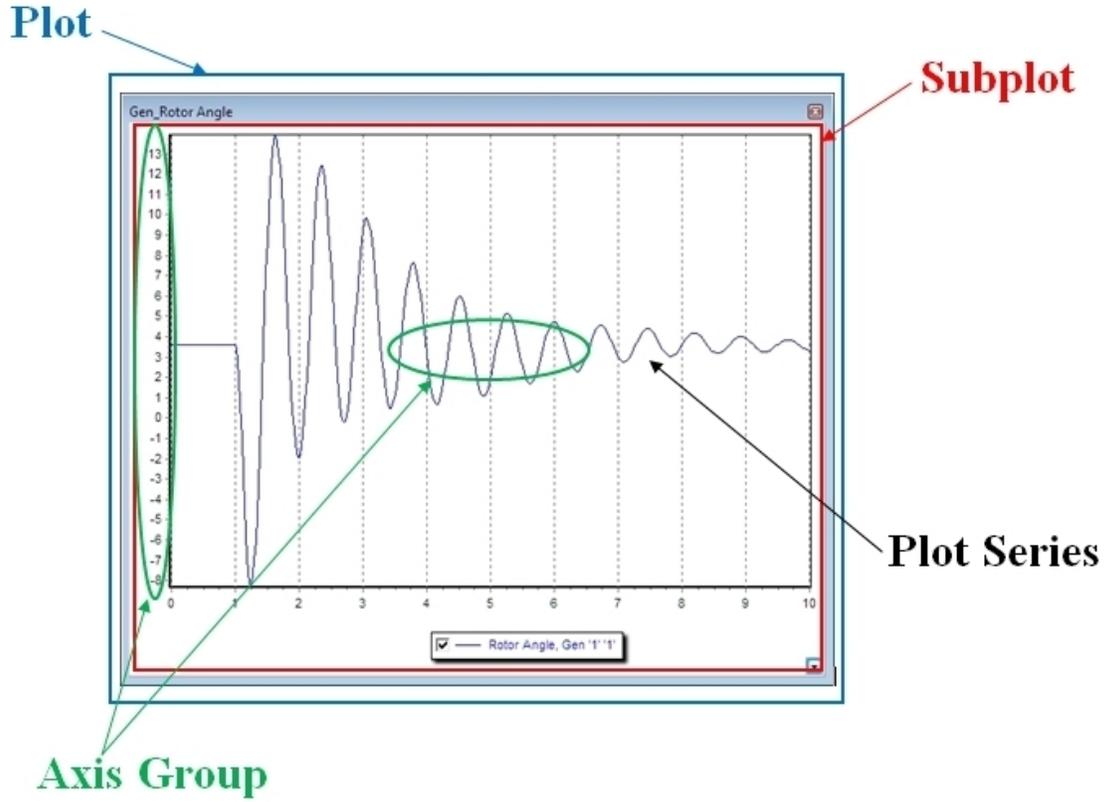
Group of plot series that share the same vertical axis. For many plots, there will only be one axis group.

Plot series

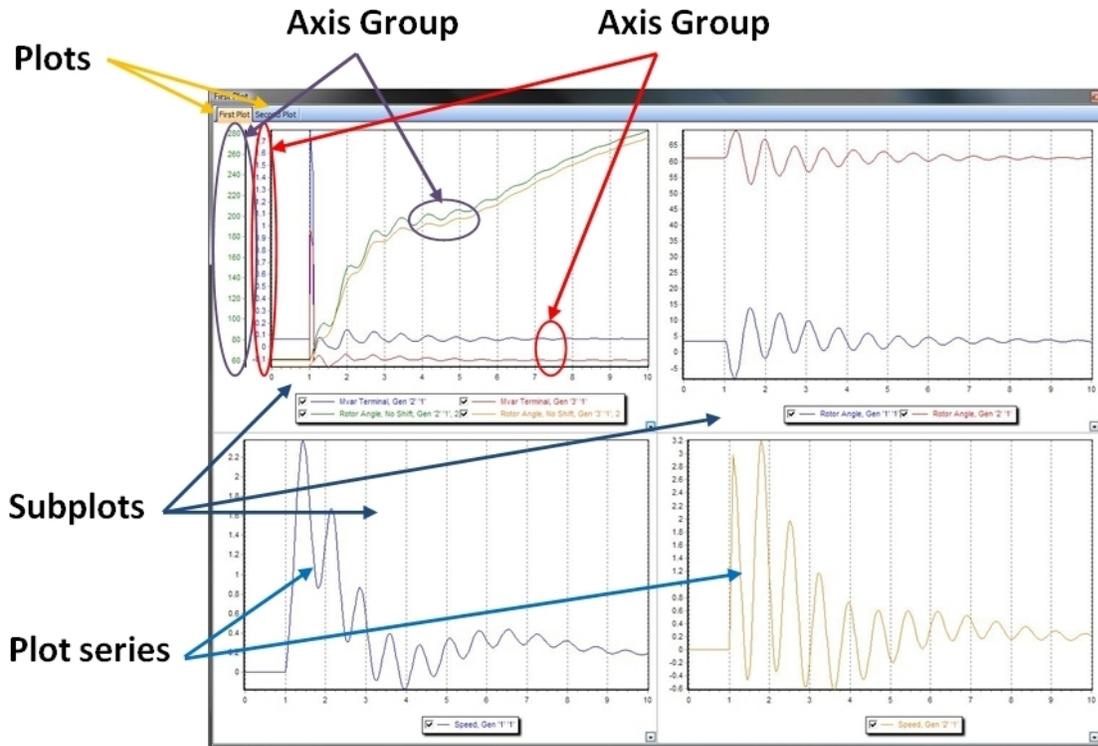
Actual graphical line representing one numerical data series.

Hierarchy of plotting: Plots contain subplots. Subplots contain axis groups. Axis groups contain plot series.

A sample plot where there is only one subplot and one axis group is shown below highlighting the key elements defined above:



A more complicated example consisting of multiple plots, subplots, axis groups, and plot series is shown below:



Navigating Around a Plot

Dragging

Right-click on the background portion of a subplot and drag with the mouse.

Zoom In

Left-click on the background portion of a subplot and drag down with the mouse.

Zoom Out

Left-click on the background portion of a subplot and drag up with the mouse.

Data Information

Left-clicking on a plot series will provide information about the selected point.

Making Plot Series Visible

Below each subplot chart there is a key listing all of the plot series on that subplot. Check and uncheck the box next to a plot series to make that plot series visible or not visible.

Advanced Options

There are a number of advanced options that are associated with a subplot. These advanced options are only accessible through a local menu available with each subplot. These advanced options can be saved or loaded from file. These options are available by right-clicking below the chart portion of a subplot, right-clicking on a plot series, or clicking the little arrow found in the bottom right-hand corner of each subplot.

Other more common options available for plots and associated components are discussed in the [Plot Designer](#) section.

The Plots page is broken into two sub-tabs that provide the means for creating and modifying plots:

Plot Designer

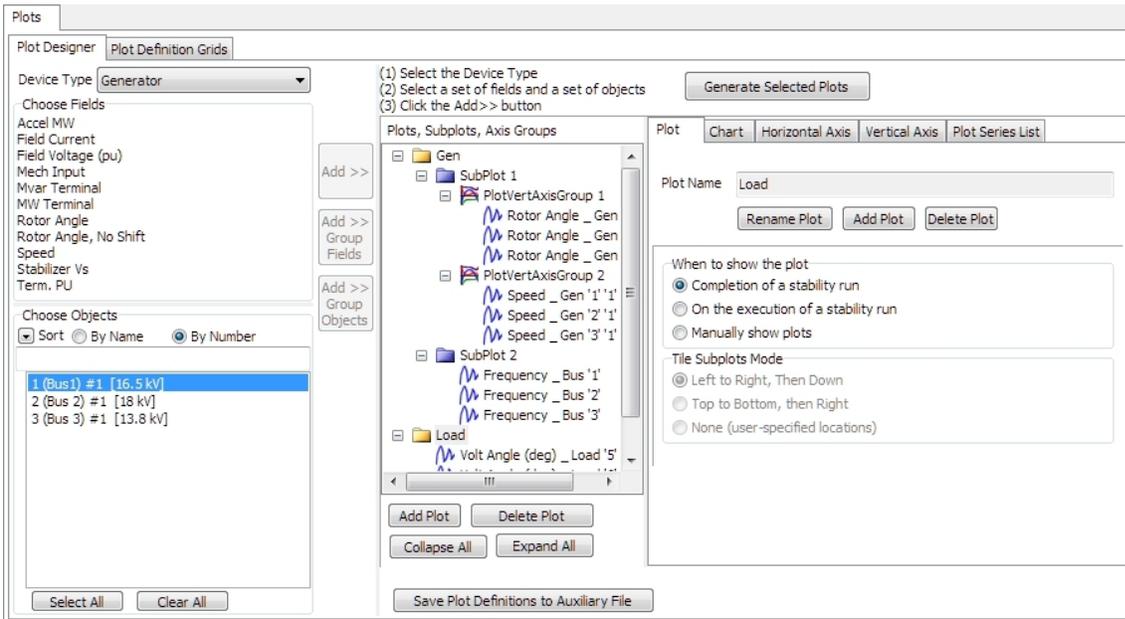
This page is used to create and modify plots.

Plot Definition Grids

This page provides summary case information displays of the various components comprising a plot: Plots, Sub Plots, Axis Groups, and Plot Series. Once plots have been created using the [Plot Designer](#), they can be modified through the Plot Definition Grids. These grids also provide the format in which plots are stored in auxiliary files.

Transient Stability Analysis Plots: Plot Designer

The Plot Designer sub-tab is found on [Plots](#) page of the [Transient Stability Analysis dialog](#). Settings on this page determine which results will be plotted and when the plots will be made visible during the transient stability analysis.



The Plot Designer page is broken into three main sections. The left side of the page allows for the selection of devices and fields that should be included in a plot. The middle section defines the plots that are created and how these should be defined in terms of subplots and axis groups. The right section of the page allows modification of all of the plot components.

Creating a New Plot

A new plot can be created in a number of ways:

- If no existing plot exists, simply selecting a **Device Type**, **Choose Fields**, and **Choose Objects** and then clicking the **Add >>**, **Add >> Group Fields**, or **Add >> Group Objects** buttons will create a new plot with a default name.
- Click the **Add Plot** button found under the **Plots, Subplots, Axis Groups** list. A prompt will appear asking for the name of the plot. Enter the name and click OK. Then use the **Device Type**, **Choose Fields**, and **Choose Objects** options and the **Add >>** buttons to add data to a plot.
- Right-click on the **Plots, Subplots, Axis Groups** list and select Add Plot. A prompt will appear asking for the name of the plot. Enter the name and click OK. Then use the **Device Type**, **Choose Fields**, and **Choose Objects** options and the **Add >>** buttons to add data to a plot.
- Click the **Add Plot** button found on the **Plot** sub-tab that is part of the **Plots, Subplots, Axis Groups** section. A prompt will appear asking for the name of the plot. Enter the name and click OK. Then use the **Device Type**, **Choose Fields**, and **Choose Objects** options and the **Add >>** buttons to add data to a plot.
- Select fields and objects in the [Results to Save](#) tables.

Device Type

Select the device type from the drop-down of all allowed types. Changing the device type will update the **Choose Fields** and **Choose Objects** accordingly.

Choose Fields

Choose the fields to be added to a plot. Multiple fields can be selected by dragging the mouse down a selection, holding down the CTRL key and selecting individual fields with the mouse, or holding down the Shift key and selecting the beginning and ending of a range with the mouse.

Choose Objects

Choose the objects to be added to a plot. Multiple fields can be selected by dragging the mouse down a selection, holding down the CTRL key and selecting individual fields with the mouse

Add >>

When clicked the selected fields and objects will be added to the appropriate plot component. Fields are selected from the **Choose Fields** list and objects are selected from the **Choose Objects** list. This button is enabled under the following conditions if at least one field and one object are selected:

Plot selected that contains only one subplot and one axis group - plot series for each field and object will be added to the selected plot in the single axis group and subplot

Subplot selected that contains only one axis group - plot series for each field and object will be added to the selected subplot in the single axis group

Axis group selected - plot series for each field and object will be added to the selected axis group

In any of these case if a plot series already exists for the selected field and object, a new plot series will not be added.

Add >> Group Fields

When clicked the selected fields and objects will be added to the appropriate plot component. Fields are selected from the **Choose Fields** list and objects are selected from the **Choose Objects** list. This button is enabled under the following conditions if at least one object and more than one field are selected:

Plot selected that contains only one subplot - axis groups will be created for each of the selected fields and plot series for each of the selected objects will be placed in each new axis group

Subplot selected - axis groups will be created in the selected subplot for each of the selected fields and plot series for each of the selected objects will be placed in each new axis group

Axis group selected - axis groups will be created in the subplot that contains the selected axis group. A new axis group will be created for each of the selected fields and plot series for each of the selected objects will be placed in each new axis group.

In any of these cases if an axis group for a selected field already exists in the corresponding subplot, a new axis group will not be created, but the plot series will be added to the existing axis group. If a plot series for a selected object already exists in the axis group, a new plot series will not be added. For an axis group to exist for a selected field, the axis group can only contain plot series for that particular field.

Add >> Group Objects

When clicked the selected fields and objects will be added to the appropriate plot component. Fields are selected from the **Choose Fields** list and objects are selected from the **Choose Objects** list. This button is enabled under the following conditions if at least one field and more than one object are selected:

Plot selected that contains only one subplot - axis groups will be created for each of the selected objects and a plot series for each of the selected fields will be placed in each new axis group

Subplot selected - axis groups will be created in the selected subplot for each of the selected objects and plot series for each of the selected fields will be placed in each new axis group

Axis group selected - axis groups will be created in the subplot that contains the selected axis group. A new axis group will be created for each of the selected objects and plot series for each of the selected fields will be placed in each new axis group.

In any of these cases if an axis group for a selected object already exists in the corresponding subplot, a new axis group will not be created, but the plot series will be added to the existing axis group. If a plot series for a selected field already exists in the axis group, a new plot series will not be added. For an axis group to exist for a selected object, the axis group can only contain plot series for that particular object.

Generate Selected Plots

Click this button to generate plots for those selected in the **Plots, Subplot, Axis Groups** list. For a plot to be selected either the plot itself or one of its components needs to be selected.

Save Plot Definitions to Auxiliary File

Click this button to save all of the plot definitions to an auxiliary file. This is useful when running analyses on power system models with common devices.

Plots, Subplots, Axis Groups

This is a listing of all plots that have been created along with associated subplots, axis groups, and data series. The list is arranged as a group of folders. Each plot is contained in its own folder. If there is more than one subplot for a plot, each subplot will be contained in a folder. Each subplot then contains a list of axis groups. If there is more than one axis group for a subplot, each axis group will be contained in a folder. Each axis group then contains a list of the plot series that it contains.

When adding devices and fields to a plot, the plot component selected in this list will determine where the data will be added when clicking one of the **Add >>** buttons. The **Add >>** buttons will be enabled according to the component selected in this list and the combination of devices and fields selected in **Choose Fields** and **Choose Objects**.

There are several buttons located at the bottom of the plot listing. These buttons affect what and how things are shown in the list.

Add Plot

Click this button to add a new plot to the list.

Delete

The caption and functionality of this button will change depending on the particular type of plot component that is selected in the list. Clicking this button will then delete the selected component and any other components that are contained in the selected component. For example, if an axis group is selected and the Delete button is clicked, all plot series contained in the axis group will be deleted along with the axis group.

Collapse All

Click this button to collapse all folders in the list.

Expand All

Click this button to expand all folders in the list.

Selecting a particular plot component (plot, subplot, axis group, plot series) from the list will change the sub-tabs that are available to the right of the list. Each of these sub-tabs allows for modifications of the different plot components as well as the components in which a particular component is contained, e.g. selecting a plot series will allow for modification of that particular series as well as the axis group in which the series is contained, the subplot in which the axis group is contained, and the plot in which the subplot is contained. Options can be changed for a plot before the plot is generated or while the plot is showing. Changing any of the options on the sub-tabs described below will update the current plot. Possible sub-tabs are as follows:

[Plot](#)

[Chart](#)

[Horizontal Axis](#)

[Vertical Axis](#)

[Plot Series List](#)

[Plot Series](#)

Transient Stability Analysis Plot Designer: Plot

The Plot sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the Plot that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. A plot is selected if any of its corresponding subplots, axis groups, or plot series are selected in the list.

Plot Name

Name of the currently selected plot.

Rename Plot

Click this button to rename the currently selected plot. A dialog box will open in which the new plot name should be entered. Click OK on this dialog to change the plot name or Cancel to abandon the change.

Add Plot

Click this button to create a new plot. A dialog will open prompting for a name for the new plot. Click OK to create the new plot or Cancel to abandon the addition.

Delete Plot

Click this button to delete the currently selected plot.

When to show the plot

Set this option to determine when a plot should be shown. A plot can automatically be shown at the completion of a stability run or on execution of a stability run. If showing on execution, the plot will be updated as the analysis progresses. If set to be shown manually, the plot must be selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) page and the **Generate Selected**

Plots button must be clicked in order to be shown. A plot can be shown manually in this manner at any time even if one of the options to show the plot automatically is selected.

Tile Subplots Mode

This option will only be enabled if a plot contains more than one subplot. This option dictates how the subplots will be placed within the plot window. If choosing **None (user-specified locations)**, the location of the subplot should be specified with the **Location** option found on the [Chart](#) sub-tab.

Transient Stability Analysis Plot Designer: Chart

The Chart sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the subplot that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. A subplot is selected if any of its corresponding axis groups or plot series are selected in the list.

The Chart portion of the subplot is the background, header, and footer of the subplot.

Subplot Number

This is an informational field. Each subplot is assigned a number when it is created. The user cannot change this value.

Add Subplot

Click this button to add a subplot to the currently selected plot.

Delete Subplot

Click this button to delete the currently selected subplot.

Visible

Check this box to make the subplot visible when its corresponding plot is shown.

Background Color

Click the color box or the **Change** button to select a new color for the chart background.

Title

Check the **Visible** box to make a chart title shown. Specify the **Font Size** and the actual title text in the box provided.

Footer

Check the **Visible** box to make a chart footer shown. Specify the **Font Size** and the actual footer text in the box provided.

Location

Set the **Top**, **Bottom**, **Left**, and **Right** location of the subplot within the plot window if choosing the option for user-specified subplot locations with the **Tile Subplots Mode** option found on the [Plot](#) sub-tab.

Advanced Options Filename

There are a number of advanced options associated with a chart that are only accessible through the local menu found on each subplot. This option allows a file containing these advanced options to be associated with the current subplot. If a file is specified, the advanced options contained in that file will be used when generating the subplot. Use the **Browse** button to browse for a file and use the **Clear** button to no longer associate a file with the current subplot. For more information about the advanced options, see the [Navigating Around a Plot, Advanced Options](#) section.

Transient Stability Analysis Plot Designer: Horizontal Axis

The Horizontal Axis sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the subplot that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. A subplot is selected if any of its corresponding axis groups or plot series are selected in the list.

The horizontal axis is shared by all plot series on the selected subplot.

Subplot Number

This is an informational field. Each subplot is assigned a number when it is created. The user cannot change this value.

Add Subplot

Click this button to add a subplot to the currently selected plot.

Delete Subplot

Click this button to delete the currently selected subplot.

Show Horizontal Axis on Plot

Check this box to show the horizontal axis on the current subplot.

Inverted

Check this box to have the scale on the horizontal axis go from high to low values when moving left to right along the axis.

Logarithmic

Check this box to use a logarithmic scale on the horizontal axis of the current subplot.

Title

Check the **Visible** box to make a horizontal axis title shown. Specify the **Font Size** and the actual title text in the box provided.

Scale

Maximum

Check the box next to Automatic to have the maximum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the maximum value rounded when the Automatic checkbox is also checked.

Increment

Specify the increment between scale markings. If zero is specified, the increment will be determined automatically.

Minimum

Check the box next to Automatic to have the minimum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the minimum value rounded when the Automatic checkbox is also checked.

Horizontal Axis Value

The horizontal axis value is normally chosen to be time. This is the default if nothing else is selected. However, the option is allowed to choose a different value for the horizontal axis.

Clear

Click this button to restore the horizontal axis value to the default of Time.

Object Plotted

Select an object to be used for the horizontal axis values.

Field Plotted

Select the field for the associated object to be used for the horizontal axis values.

Transient Stability Analysis Plot Designer: Vertical Axis

The Vertical Axis sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the axis group that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. An axis group is selected if any of its corresponding plot series are selected in the list.

The vertical axis is shared by all plot series in the selected axis group.

Axis Group Number

This is an informational field. Each axis group is assigned a number when it is created. The user cannot change this value.

Add Axis Group

Click this button to add an axis group to the currently selected subplot.

Delete Axis Group

Click this button to delete the currently selected axis group.

Show Vertical Axis on Plot

Check this box to show the vertical axis for this access group on the current subplot.

Inverted

Check this box to have the scale on the vertical axis go from high to low values when moving bottom to top along the axis.

Logarithmic

Check this box to use a logarithmic scale on the vertical axis of the current axis group.

Title

Check the **Visible** box to make the vertical axis title shown for the current axis group. Specify the **Font Size** and the actual title text in the box provided.

Scale

Maximum

Check the box next to Automatic to have the maximum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the maximum value rounded when the Automatic checkbox is also checked.

Increment

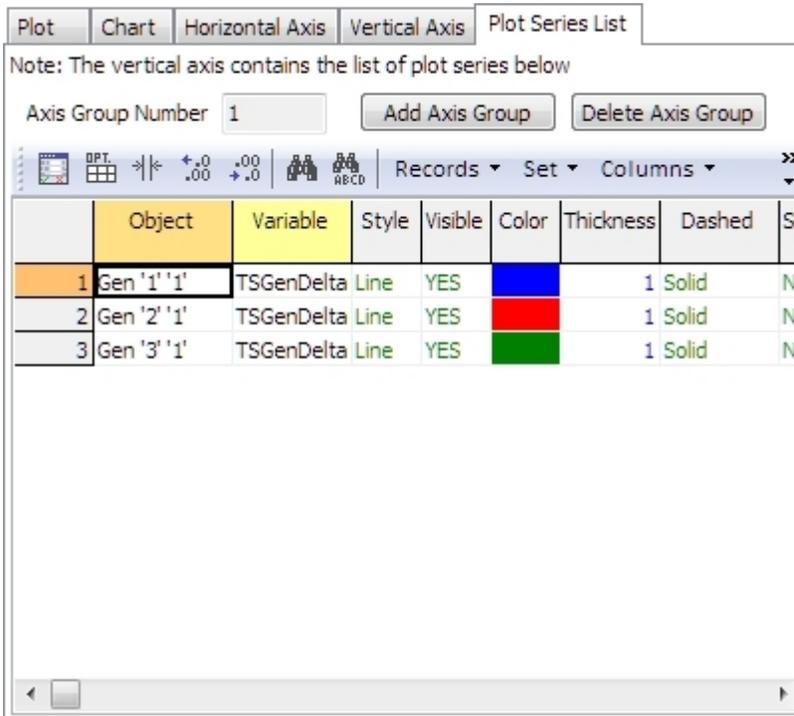
Specify the increment between scale markings. If zero is specified, the increment will be determined automatically.

Minimum

Check the box next to Automatic to have the minimum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the minimum value rounded when the Automatic checkbox is also checked.

Transient Stability Analysis Plot Designer: Plot Series List

The Plot Series List sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the plot series contained in the axis group that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab.



Axis Group Number

This is an informational field. Each axis group is assigned a number when it is created. The user cannot change this value.

Add Axis Group

Click this button to add an axis group to the currently selected subplot.

Delete Axis Group

Click this button to delete the currently selected axis group.

Plot Series List Table

This is a [case information display](#) listing all of the plot series contained in the currently selected axis group. Common options like color, thickness, and style that determine how a plot series is shown are listed in this display and can be modified here. The same fields can be modified for a single selected plot series on the [Plot Series](#) sub-tab.

Transient Stability Analysis Plot Designer: Plot Series

The Plot Series sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the plot series that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab.

Object plotted

This shows the object that is represented by the selected plot series. This is for informational purposes only and cannot be changed here.

Field plotted

This shows the field that is represented by the selected plot series. This is for informational purposes only and cannot be changed here.

Delete Plot Series

Click this button to delete the selected plot series.

Plot Series Visible

Check this box for the plot series to be shown on the plot.

Color

Click the color box or the **Change** button to select the color of the line drawn for the plot series.

Plot Series Type

Select whether the plot series should be shown by a *Line Series* or a *Point Series*. If shown by a *Point Series*, the **Point Attributes** options will be enabled allowing selection of the style and size of the points.

Line Attributes

These options will only be enabled if the **Plot Series Type** is a *Line Series*.

Style

Solid, Dashed, Dot, Dash Dot, and Dash Dot Dot are the valid options for the line style.

Thickness

Set the thickness of the line.

Stairs

Select whether the line series should be drawn with No stairs, Stair, or Inverted stair. The stair options connected successive points in a stair pattern instead of a straight line.

Symbol Every

Set this option to something other than zero if a symbol should be placed on the line for better identification. Making this value larger will make the symbol placing less dense. Use the **Point Attributes** options to define the style and size of the symbols.

Point Attributes

These options will only be enabled if the **Plot Series Type** is a *Point Series* or choosing to include symbols on a *Line Series* by setting the **Symbol Every** option to something other than zero.

Style

This determines the shape of any plot series symbols.

Height

This sets the vertical size of the plot series symbols.

Width

This sets the horizontal size of the plot series symbols.

Transient Stability Analysis: Results

The Results page is found on the [Transient Stability Analysis dialog](#). This page provides a summary of various results from a transient stability analysis run.

Results

Summary and Display Options | Minimum/Maximum Values | Time Values | Events | Plot List | Solution Process

Time Solution Started: 2/24/2009 12:51:28 PM

Time to Solve (Seconds): 0.141

Maximum Angle Difference (Deg): 77.696

Time of Max. Angle Diff. (Seconds): 1.260

Time Values Display Options

Display Angular Speed in

Deviation from Synchronous Speed

Absolute Speed

Display Angles in

Degrees

Radians

Display Angular Speed in

Rad/s

Cycle/s (hertz)

Show Time Values Grouped By

Object then Field

Field then Object

Use Area/Zone Filters

The Results page is divided into several sub-tabs:

Summary and Display Options

Time Solution Started

Time that the transient stability analysis started.

Time to Solve (Seconds)

Amount of computing time needed to solve the analysis.

Maximum Angle Difference (Deg)

Maximum angle difference between any two generator rotors at any time during the analysis.

Time of Max. Angle Diff. (Seconds)

Analysis time at which the maximum angle difference occurred.

Time Values Display Options

These options only affect how the results are shown in the table found on the **Time Values** sub-tab.

Display Angular Speed in

Angular speed can either be the Deviation from Synchronous Speed or the Absolute Speed and can be displayed in either Rad/s or Hz. If shown as the Deviation from Synchronous Speed, the angular speed should be zero until a disturbance occurs.

Display Angles in

Angles can be shown in either Degrees or Radians.

Show Time Values Grouped By

This option determines if all of the same object will be grouped together or all of the same field will be grouped together.

Use Area/Zone Filters

Check this box to filter the results by using the [Area/Zone filters](#).

Minimum/Maximum Values

This page contains tables for generators and buses listing maximum and minimum values for various parameters and the analysis time at which the values occurred.

Time Values

This page contains a table of all results selected with [Results to Save](#). The table lists values for each of the selected objects and fields for each time step.

Events

Not available yet.

Plot List

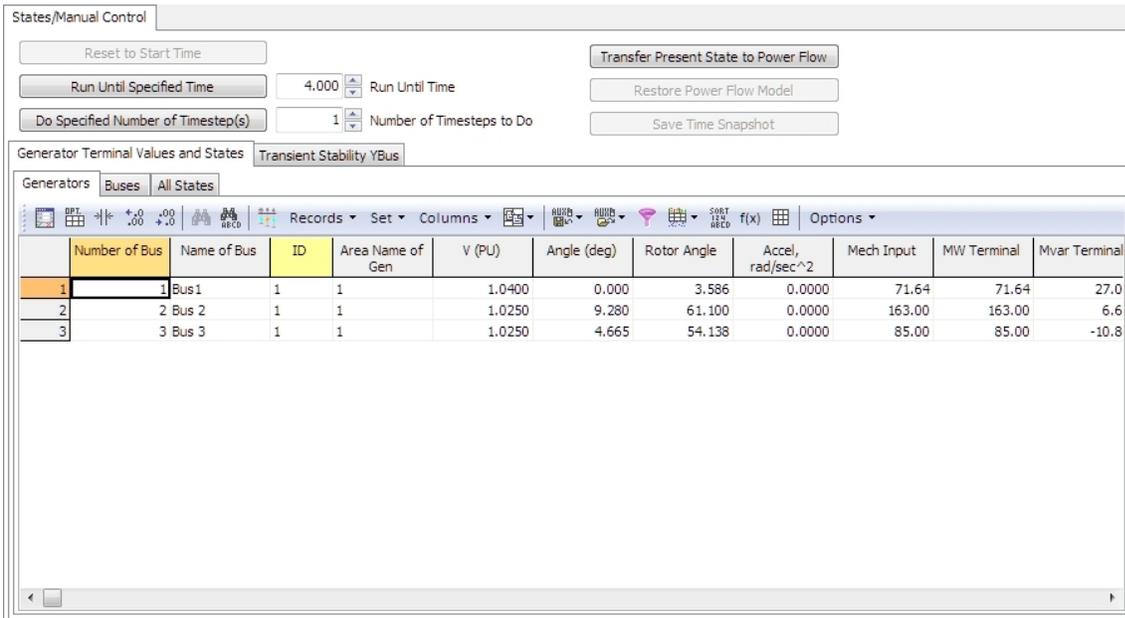
This page contains a table of all currently defined plots. A plot can be shown by selecting a plot and clicking the **Generate Selected Plots** button.

Solution Process

This page provides some summary information about the Newton solution results.

Transient Stability Analysis: States/Manual Control

The States/Manual Control page is found on the [Transient Stability Analysis dialog](#). This page provides a means to manually control a transient stability run and view generator and bus states at each step of the process.



Reset to Start Time

Click this button to reset the simulation back to the start time as specified on the [Simulation Control](#) page.

Run Until Specified Time

Click this button to run to the specified **Run Until Time**. The Run Until Time should be set in seconds. The simulation is run from the present time to this specified time. The current time is shown in the **Simulation Status** field that is displayed at the top of the [Transient Stability Analysis dialog](#). The Run Until Time must be later than the present simulation time.

Do Specified Number of Timestep(s)

Click this button to run the specified **Number of Timesteps to Do**. The simulation starts at the present time and does the number of timesteps. The current time is shown in the Simulation Status field that is displayed at the top of the [Transient Stability Analysis dialog](#).

Transfer Present State to Power Flow

Click this button to transfer the present state resulting from the transient stability analysis to the power flow model. The present state is not transferred to the power flow model when running the transient stability analysis when not using the manual mode.

Restore Power Flow Model

Click this button to restore the power flow model to the pre-transient stability analysis state. This option can be used when the transient stability state information has been transferred to the power flow model.

Save Time Snapshot

Click this button to save the power flow model to file with the present transient stability results. Transient stability results will only be saved if choosing a file type of either *.pwb or *.aux. If the transient stability state has been transferred to the power flow model, the power flow model is first restored to the pre-transient stability analysis state before saving.

Generator Terminal Values and States

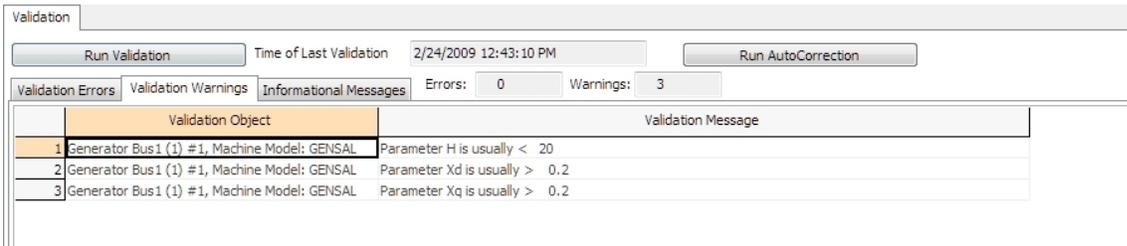
This sub-tab contains additional tabs used to display generator and bus states following the last transient stability analysis timestep.

Transient Stability YBus

This sub-tab displays the transient stability YBus following the last transient stability analysis timestep.

Transient Stability Analysis: Validation

The Validation page is found on the [Transient Stability Analysis dialog](#). This page provides a summary of any errors or warnings encountered in the transient stability models or setup and provides an option to have the software automatically correct any problems.



Run Validation

Click this button to run a validation check on the transient stability data. A validation is also run before the transient stability analysis is started.

Time of Last Validation

This shows the time that the last validation check was run. This time is either from the last time that the process was run manually or part of the check done at the beginning of a transient stability analysis run.

Run AutoCorrection

Click this button to have the software correct any errors or warnings that have been found.

Errors: and Warnings:

Provides a count of the number of errors and warnings that have been found. Errors will prevent the transient stability analysis from being run.

Summary Tables

This page is divided into three sub-tabs: **Validation Errors**, **Validation Warnings**, and **Informational Messages**. The **Validation Errors** page provides a list of any objects that have errors and provides a description of the error. Any errors will prevent the transient stability analysis from being run. The **Validation Warnings** page also provides a list of any objects that have problems and provides a description of the error. The difference is that warnings are not considered to be severe enough to prevent the transient stability analysis from being run. The **Informational Messages** page lists objects and messages about any errors or warnings that have been auto-corrected by the software.

Chapter 13: Auxiliary Script/Data Files

Auxiliary Script and Data files are used to automatically change data and control the Simulator in a batch mode fashion.

This chapter covers the following:

- Overview of Auxiliary Files
- Script Command Execution Dialog
- Quick Auxiliary Files Dialog
- Script Section
- Data Section
- Data Section for Display Auxiliary Files

Auxiliary Files

PowerWorld has incorporated the ability to import data from data sources other than power flow models into PowerWorld Simulator. Simulator has always had the ability to import supplemental data into a power flow model, but this facility was limited to ten or twelve native data formats targeted to specific applications for which the data had to have a precise format. The extensions described in this help document provide a more versatile interface to data from non-power-flow sources.

In addition to developing auxiliary data formats, a script language was also developed. The script language and auxiliary data formats are incorporated together. This format is described in this help document.

Script/Data files are called *data auxiliary* files in Simulator and typically have the file extension .AUX. These files mostly contain information about power system elements and options for running the various tools within Simulator. They do not contain any information about individual display objects contained on a one-line diagram. There are separate files called *display auxiliary* files that are available for importing/exporting display data from/to Simulator in a text format. These files are distinguished from the *data auxiliary* files by using the extension .AXD. The format for these two types of files is similar, but different object types are supported by each and require that the files be read separately. Currently, there are no script commands that are available for *display auxiliary* files.

Both file types will be generically referred to as auxiliary files. An auxiliary file may be comprised of one or more DATA or SCRIPT sections. A DATA section provides specific data for a specific type of object. A SCRIPT section provides a list of script actions for Simulator to perform.

These sections have the following format:

SCRIPT ScriptName1

```
{
script_statement_1
.
script_statement_n
}
```

DATA DataName1(object_type, [list_of_fields], file_type_specifier,create_if_not_found)

```
{
data_list_1
.
data_list_n
}
```

DATA DataName2(object_type, [list_of_fields], file_type_specifier,create_if_not_found)

```
{
data_list_1
.
data_list_n
}
```

```
}
```

SCRIPT ScriptName2

```
{  
script_statement_1  
.  
script_statement_n  
}
```

Note that the keywords SCRIPT or DATA must occur at the start of a text file line. Auxiliary files may contain more than one DATA or SCRIPT section. These sections always begin with the keyword DATA or SCRIPT. DATA sections are followed by an argument list enclosed in (). The actual data or script commands are then contained within curly braces { }.

Script Command Execution Dialog

The Script Command Execution dialog provides a location for the user to enter [script commands](#) manually, or to load [auxiliary files](#) containing [Script](#) and/or [Data](#) sections previously defined. This dialog is opened by pressing the **Script** button in the **Log** ribbon group on the [Tools](#) ribbon tab.

Loading Auxiliary Files

The first feature of the Script Command Execution dialog is that it provides a location to load previously defined [auxiliary files](#) for the currently loaded case. The **Auxiliary File** menu provides a location for loading an auxiliary file, or simply validating that the [Script](#) and/or [Data](#) sections of an Auxiliary file are correctly formatted in the file.

The **Quick Aux** option allows you to open the [Quick Auxiliary Files](#) dialog for creating a list of auxiliary files to be opened and processed en masse.

Lastly, you can export the Simulator recognized objects and object fields using the **Export Field Names** option. You can export the field names to a text for or to Excel.

Running Script Commands

The second feature of the Script Command Execution dialog is that you can run script commands manually. To run a script command, type the command into the display, and press the **Execute** button. Note that if the **Execute on ENTER key** option is checked, the command will also be processed when ENTER is pressed. Note that similar to script command syntax in the [Script](#) section of auxiliary files, script commands must be ended with a semi-colon (;) in the Script Command Execution dialog as well.

You can enter multiple script commands to be processed in sequence in this display. To do so, you must uncheck the option **Execute on ENTER key**. Then you can press enter after each script command to move to the next line and enter another command. Use the **Execute** button to process the sequence of script commands.

If you are running a sequence of script commands and wish to abort the run, use the **Abort** button.

If you wish to view the message log while script commands are processing, open the log using the **Show Log** button.

Quick Auxiliary Files Dialog

The Quick Auxiliary Files dialog can be accessed from the **Quick Aux Files** menu option of the [Script Command Execution](#) dialog. This dialog gives you a location for creating a list of auxiliary files to be processed en masse for the currently loaded case.

Using the **Define** option of the **Quick Aux Files** menu, the Quick Auxiliary Files dialog will open. Use this dialog to **Add** previously defined [auxiliary files](#) to the list of files to be processed. You can rearrange the order of the files by selecting a file and using the up and down arrows on the right to move the selected file within the list, or you can sort them alphabetically using the **Sort** button. To remove an auxiliary file from the list, use the **Delete** button.

Once the list of auxiliary files to process is complete, press the **Execute** button to process the list of auxiliary files.

Script Section

The SCRIPT section begins with a left curly brace and ends with a right curly brace.

SCRIPT ScriptName

```
{
script_statement_1
script_statement_2
.
.
.
script_statement_n
}
```

Scripts may optionally contain a ScriptName. This enables you to call a particular SCRIPT by using the [LoadScript action](#). After the optional name, the SCRIPT section begins with a left curly brace and ends with a right curly brace. Inside of this, script statements can be given.

In general, a script statement has the following format:

Keyword(arg1, arg2, ...);

- Statement starts with a keyword.
- The keyword is followed by an argument list which is encompassed in parentheses ().
- The arguments are separated by commas.
- If a single argument is a list of things, this list is encompassed by braces []. (eg. SetData)
- Statements end with a semicolon.
- Statements may take up several lines of the text file.
- You may put more than one statement on a single text line.

Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language, but with the addition of a **submode** feature. Submodes limit what script commands can be called. Only those commands available to the current submode can be executed. You will always be in one of the submodes when executing a script. If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to the **EDIT, CASE** submode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to the **RUN, POWERFLOW** submode.

To switch submodes, use the [EnterMode \(mode or submode\)](#) script command. Available submodes include:

Edit Mode

Case Submode

Run Mode

PowerFlow Submode

Contingency Submode

ATC Submode

Fault Submode
PV Submode
QV Submode

Script General Actions

The following actions are available to you regardless of the Mode or SubMode:

```

RenameFile    ("oldfilename",    "newfilename");
CopyFile      ("oldfilename",    "newfilename");
DeleteFile    ("filename");
LoadAux       ("filename",      CreateIfNotFound);
LoadScript    ("filename",      ScriptName);

LoadData      ("filename",      DataName,      CreateIfNotFound);

SelectAll                                (objecttype,      filter);
UnselectAll                                (objecttype,      filter);
Delete                                              (objecttype,      filter);
DeleteIncludingContents (objecttype,      filter);

SaveData("filename",filetype,objecttype,[fieldlist],[subdatalist],filter,[SortFieldList]);
SaveDataWithExtra("filename",filetype,objecttype,[fieldlist],[subdatalist],filter,[SortFieldList],[Header_List],[Header_Value_List]);
SaveDataUsingExportFormat("filename",filetype,FormatName,ModelToUse);

SetData      (objecttype,      [fieldlist],      [valuelist],      filter);

CreateData    (objecttype,      [fieldlist],      [valuelist]);

ChangeData    (objecttype,      [fieldlist],      [valuelist],      filter);      (NOT
                                                    AVAILABLE
                                                    YET)

SaveObjectFields ("filename",      objecttype,      [fieldlist]);

WriteTextToFile ("filename",      "text...");
SetCurrentDirectory ("filedirectory",      CreateIfNotFound);
ExitProgram;
NewCase;
OpenCase      ("filename");      // assumes to open
                                as PWB
OpenCase      ("filename",      openfiletype);
                                PWB, GE, PTI, or
                                CF
SaveCase      ("filename");      // assumes to save
                                as PWB
SaveCase      ("filename",      savefiletype);
                                PWB, PWB5,
                                PWB6,
                                PWB7,PWB8,
                                PWB9, PWB10,
                                PWB11, PWB12,
                                PWB13
                                PTI23, PTI24,

```

```

EnterMode          (mode or
                   EDIT
                   RUN
                   PTI25, PTI26,
                   PTI27,PTI28,
                   PTI29, PTI30
                   GE, GE14, GE15
                   CF
                   submode);
                   CASE
                   POWERFLOW
                   CONTINGENCY
                   ATC
                   FAULT
                   PV
                   QV

LogClear;
LogSave           ("filename",      AppendFile);
                   YES or NO

LogAdd            ("string...");

LogAddDateTime   ("label",      IncludeDate,      IncludeTime,      IncludeMilliseconds);
                   YES or NO      YES or NO          YES or NO

CaseDescriptionClear;
CaseDescriptionSet ("text...",      Append
                   YES or NO

SaveYbusInMatlabFormat ("filename",      IncludeVoltages);
                   YES or NO

SaveJacobian     ("Jacfilename",      "JIDfilename",      filetype,      JacForm);
                   M
                   R
                   TXT          P

SetParticipationFactors (Method,      ConstantValue,      Object);
                   MAXMW RAT
                   RESERVE
                   CONSTANT      value
                   [Area num]
                   [Zone num]
                   SYSTEM
                   AREAZONE or
                   DISPLAYFILTERS

GenForceLDC_RCC      (filter);

CalculateRXBGFromLengthConfigCondType (filter);

DirectionsAutoInsert (Source,      Sink,      Delete
                   AREA          AREA          Existing,
                   ZONE          ZONE          YES or
                   INJECTION      INJECTION  NO
                   GROUP          GROUP
                   SLACK

DeterminePathDistance ([start],      BranchDistMeas,      BranchFilter,      BusField);

```

DetermineShortestPath ([start], [end], BranchDistMeas, BranchFilter, FileName);
 ChangeSystemMVABase (NewBase);

RenameFile("oldfilename", "newfilename");

Use this action to rename a file from within a script.

"oldfilename": The present file name.

"newfilename": The new file name desired.

CopyFile("oldfilename", "newfilename");

Use this action to copy a file from within a script.

"oldfilename": The present file name.

"newfilename": The new file name desired.

DeleteFile("filename");

Use this action to delete a file from within a script.

"filename": The file name to delete.

LoadAux("filename", CreateIfNotFound);

Use this action to load another auxiliary file from within a script.

"filename": The filename of the auxiliary file being loaded.

CreateIfNotFound: Set to YES or NO. YES means that objects that cannot be found will be created while reading in DATA sections from filename. If this parameter is not specified, then NO is assumed.

LoadScript("filename", ScriptName);

Use this action to load a named Script Section from another auxiliary file. This will open the auxiliary file denoted by "filename", but will only execute the script section specified.

"filename": The filename of the auxiliary file being loaded.

ScriptName: The specific ScriptName from the auxiliary file which should be loaded.

LoadData("filename", DataName, CreateIfNotFound);

Use this action to load a named Script Section from another auxiliary file. This will open the auxiliary file denoted by "filename", but will only execute the script section specified.

"filename": The filename of the auxiliary file being loaded.

DataName: The specific ScriptName from the auxiliary file which should be loaded.

CreateIfNotFound: Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections from filename. If this parameter is not specified, then NO is assumed.

SelectAll(objecttype, filter);

Use this to set the selected property of objects of a particular type to true. A filter may optionally be specified to only set this property for objects which meet a filter.

objecttype: The objecttype being selected.

filter: There are three options for the filter:

SelectAll(objecttype);: No filter specified means to select all objects of this type.

SelectAll(objecttype, "filtername");: "filtername" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SelectAll(objecttype, AREAZONE);: AREAZONE means select those that meet the area/zone filters.

UnSelectAll(objecttype, filter);

Same as SelectAll, but this action sets the Selected field to false.

Delete(objecttype, filter);

Use this to delete objects of a particular type. A filter may optionally be specified to only delete objects that meet a filter.

objecttype: The objecttype being selected.

filter: There are four options for the filter:

Delete(objecttype);: No filter specified means to delete all objects of this type.

Delete(objecttype, "filtername");: "filtername" means delete those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

Delete(objecttype, AREAZONE);: AREAZONE means delete those that meet the area/zone filters.

Delete(objecttype, SELECTED);: SELECTED means delete those objects whose Selected field is set to *YES*.

DeleteIncludingContents(objecttype, filter);

Use this to delete objects of a particular type and other objects that these contain. Currently, only multi-section lines (objecttype = MultiSectionLine) can be used with this command. The branches and dummy buses that belong to multi-section lines will also be deleted along with the multi-section lines. A filter may optionally be specified to only delete objects that meet a filter.

objecttype: The objecttype being selected (presently we only support objecttype = MultiSectionLine).

filter: There are four options for the filter:

DeleteIncludingContents(objecttype);: No filter specified means to delete all objects of this type.

DeleteIncludingContents(objecttype, "filtername");: "filtername" means delete those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

DeleteIncludingContents(objecttype, AREAZONE);: AREAZONE means delete those that meet the area/zone filters.

DeleteIncludingContents(objecttype, SELECTED);: SELECTED means delete those objects whose *selected* field is set to YES.

SaveData("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList]);

Use this action to save data in a custom defined format. A filter may optionally be specified to save only objects which meet a filter, and a sort order list may also optionally be specified that will allow saving the objects in a particular order.

"filename": The file to save the data to

filetype:

AUX (or AUXCSV): save as a space-delimited (or comma-delimited) auxiliary data file.

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

objecttype: The objecttype being saved.

[fieldlist]: A list of fields to save. For numeric fields, the number of digits and the number of decimal places (digits to right of decimal) can be specified by using the following format for the field, *variablename:location:digits:rod*.

[subdatalist]: A list of the subdata objecttypes to save with each object record.

filter: There are four options for the filter:

SaveData(...);: No filter specified means to save all objects of this type.

SaveData(..., "filtername");: "filtername" means save those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SaveData(..., AREAZONE);: AREAZONE means save those that meet the area/zone filters.

SaveData(..., SELECTED);: SELECTED means save those objects whose *selected* field is set to YES.

[SortFieldList]: This allows the specification of a sort order in which the data will be saved. The format is:

[variablename1:+:0, variablename2:-:1]

where

variablename is the name of the field to sort by. There is no limit to how many fields can be specified for sorting. For fields that require a location other than zero, *variablename* can be in the format **fieldname:location**.

+ or **-** for the second parameter indicates sort ascending for **+** and sort descending for **-**. This parameter must be specified.

0 or **1** for the third parameter indicates: **0** - case insensitive and do not use absolute value, **1** - case sensitive or use absolute value. This parameter is optional.

SaveDataWithExtra("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList], [Header_List], [Header_Value_List]);

Use this action to save data in a custom defined format. User-specified fields and field values can also be specified in the output. Optional parameters are filter, SortFieldList, Header_List, and Header_Value_List.

"filename": The file to save the data to

filetype:

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

CSVNOHEADER: save as a normal CSV text file, without the AUX file formatting. The object name and field names are NOT included. This option is useful when appending data of the same object type and field list into a common file.

Data cannot be saved using the AUX filetype with this command.

objecttype: The objecttype being saved.

[fieldlist]: A list of fields to save. For numeric fields, the number of digits and the number of decimal places (digits to right of decimal) can be specified by using the following format for the field, `variablename:location:digits:rod`.

[subdatalist]: A list of the subdata objecttypes to save with each object record.

filter: There are four options for the filter:

SaveData(...);: No filter specified means to save all objects of this type.

SaveData(..., "filtername");: "filtername" means save those that meet the filter.

SaveData(..., AREAZONE);: AREAZONE means save those that meet the area/zone filters.

SaveData(..., SELECTED);: SELECTED means save those objects whose *selected* field is set to YES.

[SortFieldList]: This allows the specification of a sort order in which the data will be saved. The format is:

[variablename1:+:0, variablename2::-:1]

where

variablename is the name of the field to sort by. There is no limit to how many fields can be specified for sorting. For fields that require a location other than zero, `variablename` can be in the format **fieldname:location**.

+ or **-** for the second parameter indicates sort ascending for **+** and sort descending for **-**. This parameter must be specified.

0 or **1** for the third parameter indicates: **0** - case insensitive and do not use absolute value, **1** - case sensitive or use absolute value. This parameter is optional. If it is not specified, the default is 0.

[Header_List]: This allows the specification of user-defined fields that will appear in the output. Headers should be specified as a list of comma delimited strings. A string should be enclosed in double quotes if the string contains a comma. Header strings cannot be blank.

[Header_Value_List]: Allows the specification of the values that should be assigned to the user-defined fields specified by Header_List. Specifying the values is optional. If specified, there must be as many values specified as there are headers. If not specified, all values are blank. Each object will use the same specified value for the specified field. To use different values for different objects and save these in the same file, make use of the CSVNOHEADER file format and filtering.

SaveDataUsingExportFormat("filename", filetype, FormatName, ModelToUse);

Use this action to save data in a user-defined format that has previously been defined.

"filename": The file to save the data to

filetype:

AUX (or AUXCSV): save as a space-delimited (or comma-delimited) auxiliary data file.
 CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

FormatName: The name of the Object Export Format Description to use.

ModelToUse: Optional parameter that indicates the model to use.

FULL: Full-topology model. This is the default if the parameter is omitted.

CONSOLIDATED: Consolidated planning-type model. This option will only work with the Topology Processing add-on.

SetData(objecttype, [fieldlist], [valuelist], filter);

Use this action to set fields for particular objects. If a filter is specified, then it will set the respective fields for all objects which meet this filter. Otherwise, if no filter is specified, then the keyfields must be included in the field list so that the object can be found.

objecttype: The objecttype being set.

[fieldlist]: A list of fields that you want to save.

[valuelist]: A list of values to set the respective fields to.

filter: There are four options for the filter:

SetData(...);: No filter specified: set data only for the object described by the [fieldlist] and [valuelist] parameters.

SetData(..., "filtername");: "filtername": set data for all objects that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SetData(..., AREAZONE);: AREAZONE: set data for all objects that meet the area/zone filters.

SetData(..., SELECTED);: SELECTED: set data for all objects whose *selected* field is set to YES.

CreateData(objecttype, [fieldlist], [valuelist]);

Use this action to create particular objects. Note that the key fields for the objecttype must be specified.

objecttype: The objecttype being set.

[fieldlist]: A list of fields that you want to save.

[valuelist]: A list of values to set the respective fields to.

SaveObjectFields("filename", objecttype, [fieldlist]);

Use this action to save a list of fields available for the specified objecttype to a CSV file. Format of the file is *variablename, field, col header, description*.

"filename": The file path and name to save.

objecttype: The type of object for which fields should be saved.

[fieldlist]: List of fields for which information will be saved.

WriteTextToFile("filename", "text...");

Use this action to write text to a file. If the specified file already exists, the text will be appended to the file. Otherwise, it creates the file and writes the text to the file.

"filename": The file path and name to save.

"text...": The text to be written to the file.

SetCurrentDirectory("filedirectory", CreateIfNotFound);

Use this action to set the current working directory.

"filedirectory": The path of the work directory.

CreateIfNotFound: Set to YES or NO. YES means that if the directory path can not be found, the directory will be created. If this parameter is not specified, then NO is assumed.

ExitProgram;

Immediately exits the program with no prompts.

NewCase;

This action clear out the existing case and open a new case from scratch.

OpenCase("filename", OpenFileType);

This action will open a case stored in "filename" of the type OpenFileType;

"filename": The file to be opened.

OpenFileType: An optional parameter saying the format of the file begin opened is. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PTI, GE, CF, AUX.

SaveCase("filename", SaveFileType);

This action will save the case to "filename" in the format SaveFileType.

"filename": The file name to save the information to.

SaveFileType: An optional parameter saying the format of the file to be saved. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PWB5, PWB6, PWB7, PWB8, PWB9, PWB10, PWB11, PWB12, PWB13

PTI (means PTI30), PTI23, PTI24, PTI25, PTI26, PTI27, PTI28, PTI29, PTI30

GE (means GE15), GE14, GE15

CF

AUX

EnterMode(mode or submode);

This action will tell Simulator what mode or submode to enter before performing the next script command.

SubMode: The submode to enter. A parameter stating what submode to put the program in. Options available are CASE, POWERFLOW, CONTINGENCY, ATC, FAULT, PV, QV. One may also put in RUN or EDIT which will place the program in the POWERFLOW or CASE respectively.

LogClear;

Use this action to clear the Message Log.

LogSave("filename", AppendFile);

This action saves the contents of the Message Log to "filename".

"filename": The file name to save the information to.

AppendFile: Set to YES or NO. YES means that the contents of the log will be appended to "filename". NO means that "filename" will be overwritten.

LogAdd("string...");

Use this action to add a personal message to the MessageLog.

"string...": The string that will appear as a message in the log.

LogAddDateTime("label", IncludeDate, IncludeTime, IncludeMilliseconds);

Use this action to add a personal message to the MessageLog, including the posting date/time.

"label": A string that will appear at the start of the line containing the date/time.

IncludeDate: Set to YES or NO. YES means that the date will be added after the string specified. If this parameter is not specified, then YES is assumed.

IncludeTime: Set to YES or NO. YES means that the time will be added after the date. If there's no date, then it will be added after the string specified. If this parameter is not specified, then YES is assumed.

IncludeMilliseconds: Set to YES or NO. YES means that the time including milliseconds will be added after the date or string specified. This is valid only if the IncludeTime is set to YES. If this parameter is not specified, then YES is assumed.

CaseDescriptionClear;

Use this action to clear the case description of the presently open case.

CaseDescriptionSet("text...", Append);

Use this action to set or append text to the case description

"text...": Specify the text to set/append to the case description

Append: YES will append the text to the existing case description. NO will replace the case description

SaveYbusInMatlabFormat("filename", IncludeVoltages);

Use this action to store the power system bus admittance matrix (Ybus) in a text format that can be easily read into other programs such as MATLAB®.

"filename": The file name to save the information to. The Ybus data is stored using the MATLAB sparse matrix format in the matrix Ybus.

IncludeVoltages: Set to YES or NO. YES means that the bus voltages are also stored but in the vector V.

SaveJacobian("filename", IncludeVoltages);

Use this action to store the power flow Jacobian in a text format that can be easily read into other programs such as MATLAB®.

"Jacfilename": The file name to save the Jacobian data to. The Jacobian is stored using the MATLAB sparse matrix format in the matrix Jac.

"JIDfilename": The file name to save the text identifier information to. This information is used to translate the bus numbering convention used in the Jacobian and Ybus files with the actual bus number and name in the case.

filetype: A parameter saying the format of the file to be saved. It may be one of the following strings:

M: MATLAB .M Format.

TXT: ASCII format.

JacForm: Specifies the form of the Jacobian. It may be one of the following strings:

R: Rectangular form

P: Polar form.

SetParticipationFactors(Method, ConstantValue, Object);

Use this action to define the participation factors for multiple generators.

Method: The formula used to calculate the participation factors for each generator. It may be one of the following strings:

MAXMW RAT: The participation factor for each generator is set to the generator's maximum MW capability.

RESERVE: The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute (Max MW rating - Present MW).

CONSTANT: The participation factor for each generator is set to the same specified value.

ConstantValue: The value used if CONSTANT method is specified. If CONSTANT method is not specified, enter 0 (zero).

Object: The set of generators to which the participation factors will be assigned. It may be one of the following settings:

[Area num], [Area "name"] [Area "label"]: For all the generators in area

[Zone num], [Zone "name"] [Zone "label"]: For all the generators in zone

SYSTEM: For all the generators in the system regardless of their area or zone affiliation.

AREAZONE or DISPLAYFILTERS: For just the generators whose area/zone/owner filter criteria evaluates to true.

GenForceLDC_RCC(filter);

Use this action to convert the voltage setpoint for Line Drop and Reactive Current Compensation (LDC/RCC) for multiple generators.

filter: There are four options for the filter:

GenForceLDC_RCC;: No filter specified means to convert voltage setpoint for LDCC/RCC on all the generators of the system.

GenForceLDC_RCC ("filename"); "filename" means to convert voltage setpoint for LDCC/RCC on those generators that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filename" then takes on the form "<objecttype>objecttype's filename".

GenForceLDC_RCC (AREAZONE); AREAZONE means to convert voltage setpoint for LDCC/RCC on those generators that meet the area/zone filters.

GenForceLDC_RCC (SELECTED); SELECTED means to convert voltage setpoint for LDCC/RCC on those generators whose *selected* field is set to YES.

CalculateRXBGFromLengthConfigCondType(filter);

Use this action to go through branches in the power system and automatically recalculate the per unit R, X, G, and B values using the TransLineCalc tool. The branches Conductor Type, Tower Configuration, and Line Length will be passed to the TransLineCalc tool and new R, X, G and B values will be calculated. This is only available if you have installed the TransLineCalc tool.

filter: There are four options for the filter which operate identically to the option for the GenForceLDC_RCC action above.

CalculateRXBGFromLengthConfigCondType; No filter specified means to perform calculation on all branches.

CalculateRXBGFromLengthConfigCondType ("filename"); "filename" means to perform calculation on those branches that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filename" then takes on the form "<objecttype>objecttype's filename".

CalculateRXBGFromLengthConfigCondType; AREAZONE means to perform calculation on those branches that meet the area/zone filters.

CalculateRXBGFromLengthConfigCondType: SELECTED means to perform calculation on those branches whose *selected* field is set to YES.

DirectionsAutoInsert(Source, Sink, DeleteExisting, UseDisplayFilters, Start, Increment);

Use this action to automatically insert multiple directions for PTDF studies.

Source, Sink: The type of object used as the source/sink of the directions. Currently, only the following types of directions can be automatically defined:

AREA SLACK: Directions from areas to slack bus of the system.

ZONE SLACK: Directions from zones to slack bus of the system.

INJECTIONGROUP SLACK: Directions from injection groups to slack bus of the system.

AREA AREA: Directions from areas to other areas of the system.

ZONE ZONE: Directions from zones to other zones of the system.

INJECTIONGROUP INJECTIONGROUP: Directions from injection groups to other injection groups of the system.

DeleteExisting: Set to YES or NO. YES means that previously defined directions will be deleted before the new directions are automatically inserted. NO means automatically inserted directions will be added to the list of previously defined directions. If this parameter is not specified, then YES is assumed.

UseDisplayFilters: Set to YES or NO. YES means that only Areas and Zones with their Area/Zone filters set to YES will be used when automatically inserting directions. NO specifies all

Areas and Zones will be used to automatically insert directions regardless of their Area/Zone filters. If this parameter is not specified, then NO is assumed.

Start: Integer value that specifies the first number used when automatically numbering the automatically inserted directions. If this parameter is not specified, then 1 is assumed.

Increment: Integer value that will be used as increment in numbering the subsequent automatically inserted directions. If this parameter is not specified, then 1 is assumed.

DeterminePathDistance([start], BranchDistMeas, BranchFilter, BusField);

Use this action to calculate a distance measure at each bus in the entire model. The distance measure will represent how far each bus is from the starting group specified. The distance measure can be related to impedance, geographical distance, or simply the number of nodes.

[start] : is an the starting place which is enclosed in brackets. The starting place may be either an Area, Zone, SuperArea, SUBstation, Injection Group, or Bus.

BranchDistMeas : is either X, Z, Length, Nodes, or a variable name for a branch.

X : means use the series reactance,

Z : means use $\sqrt{X^2 + R^2}$,

Length : means us the Length field, and

Nodes : means treat each branch as a length of one.

variablename : Otherwise use any Branch object variable name.

BranchFilter : is either All, Selected, Closed or the name of a branch Advanced Filter. This parameter is used to specify which branch can be traversed at all.

All : means all branches can be traversed

Selected : means only branch whose Selected field is YES can be traversed

Closed : means only branches that are CLOSED can be traversed.

Filtername : means only branches who meet the Advanced Filter named can be traversed. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

BusField : is the variablename of a Bus field. This field is populated with the minimum distance from the Start Place to that bus. All buses in the start group will have a distance measure of zero. Buses which can not be reached from the start group will have a distance measure of -1.

DetermineShortestPath([start], [end], BranchDistanceMeasure, BranchFilter, Filename);

Use this action to calculate the shortest path between a starting group and an ending group. The results will be written to a textfile specified by filename. In the text file, the first bus listed will be in the end grouping and the last bus listed will be the start grouping. The result text file will have a line for each bus passed. Each line will contain three entries delimited by a space: "Number DistanceMeasure Name".

[start] : same as the starting place for the DeterminePathDistance script command

[end] : same as the starting place for the DeterminePathDistance script command

BranchDistanceMeasure : same as for DeterminePathDistance script command

BranchFilter : same as for DeterminePathDistance script command

Filename : is a filename (may need to be enclosed in quotes) to which the results will be written.

ChangeSystemMVABase(NewBase);

Use this action to change the system MVA base to the specified value and update all internal data structures to store values on the new base.

NewBase: New power system base in MVA.

Script Online Actions

OpenOneLine	("filename",	"view",	FullScreen,	ShowFull);
CloseOneLine	("OnlineName");			
SaveOnline	("filename",	"OnlineName",	SaveFileType);	
ExportOnlineAsShapeFile	("filename",	"OnlineName",	"ShapeFileExport DescriptionName",	UseLonLat);
				YES or NO
LoadAXD	("filename",	"OnlineName",	CreateIfNotFound);	

OpenOnline("filename", "view", FullScreen);

Use this action to open a online diagram. This can only be used in SimAuto and cannot be used to actually view a online. This action is most useful when wanting to save a online diagram in a different file format.

"filename": The file name of the online diagram to open.

"view": The view name that should be opened. Pass an empty string to denote no specific view.

FullScreen: Set to YES or NO. YES means that the online diagram will be opened in full screen mode. If this parameter is not specified, then NO is assumed.

ShowFull: Optional parameter. Set to YES to open the online and apply the Show Full option. Set to NO to open the online and leave the online as is. Default is NO if not specified.

CloseOnline("OnlineName");

Use this action to close an open online diagram without saving it. If the name is omitted, the last focused online diagram will be closed.

"OnlineName": The name of the online diagram to close.

SaveOnline("filename", "OnlineName", SaveFileType);

Use this action to save an open online diagram to a file.

"filename": The path and file name of the file to save.

"OnlineName": Name of the open online to save.

SaveFileType: Type of file to save. Valid options are PWB, PWB5, PWB6, PWB7, PWB8, PWB9, PWB10, PWB11, PWB12, PWB13. If omitted, PWB, which is the most recent version, will be assumed.

ExportOnlineAsShapeFile("filename", "OnlineName", "ShapeFileExportDescriptionName", UseLonLat);

Use this action to save an open online diagram to a shapefile.

"filename": The file name of the shapefile to save.

"OnlineName": The name of the online diagram to save to a shapefile. The online diagram must be open. Use the OpenOnline script command if necessary to open the appropriate online.

"ShapeFileExportDescriptionName": Name of the ShapeFile Export Description to use when saving the shapefile.

UseLonLat: Set to YES or NO. YES means that the coordinates of objects on the oneline diagram will be saved using longitude,latitude. This will only be true if a valid map projection is in use with the oneline diagram. Otherwise, the coordinates will be saved in x,y. If this parameter is set to NO, the coordinates will be saved in x,y. If this parameter is not specified, YES is assumed.

LoadAXD("filename", "OnelineName", CreateIfNotFound);

Use this action to apply a display auxiliary file to an open oneline diagram. This action is only allowed when using SimAuto.

"filename": The file name of the display auxiliary file to load.

"OnelineName": The name of the oneline diagram to which to apply the display auxiliary file. Use the OpenOneline script command if necessary to open the appropriate oneline. If the specified oneline is not open, a new one will be created with the given name.

CreateIfNotFound: Set to YES or NO. YES means that objects which cannot be found will be created while reading in DATA sections from filename.

Script Edit Mode Actions

The following script commands are available during the Case submode of Edit mode:

```

Equivalence;
DeleteExternalSystem;

SaveExternalSystem    ("filename",    Savefiletype,    withties);

Scale    (scaletype,    basedon,    [parameters],    ScaleMarker);
         LOAD           MW           [P,Q]          BUS
         GEN           FACTOR        [P] (means    AREA
                                       constant pf)

         INJECTIONGROUP           ZONE
         BUSSHUNT
Move     ([elementA],    [P, +Q, -Q]
         [destination
         parameter]);
         [GEN numA idA]    [numB idB]
         [LOAD numA idA]  [numB idB]
         [BRANCH numA1 numA2 cktA] [numB1
         numB2 cktB]

Combine  ([elementA],    [elementB]);
         [GEN numA      [GEN numB idB]
         idA]
         [LOAD          [LOAD numB idB]           // NOT
         numA idA]      AVAILABLE
         YET
         [BRANCH        [BRANCH numB1 numB2 cktB] // NOT
         numA1          AVAILABLE
         numA2          YET
         cktA]

SplitBus ([element],    NewBusNumber,    InsertBusTieLine,    LineOpen);
         [BUS num]      num           YES or NO          YES or NO

MergeBuses    (NewBusNumber,    filter);
              num

TapTransmissionLine ([element],    PosAlongThe    NewBusNumber,    ShuntModel,    TreatAs
                    [BRANCH    Line,           num,           LINESHUNTS     MSLine);
                    numA1      Value in %     YES or NO      YES or
                    numA2      NO
                    cktA];
                    CAPACITANCE

InterfacesAutoInsert (Type,    DeleteExisting,    UseDisplayFilters,    Prefix,    Limits);
                    AREA     YES or NO          YES or NO          "string"    AUTO
    
```

ZONE

ZEROS
[value1,
... value
8]**Equivalence**

This action will equivalence a power system. All options regarding equivalencing are handled by the Equiv_Options objecttype. Use the SetData() action, or a DATA section to set these options prior to using the Equivalence() action. Also, remember that the property BusEquiv must be set true for each bus that you want to equivalence.

DeleteExternalSystem

This action will delete part of the power system. It will delete those buses whose property BusEquiv must is set true.

SaveExternalSystem("filename", SaveFileType, WithTies)

This action will save part of the power system to a "filename". It will save only those buses whose property BusEquiv must is set true.

"filename": The file name to save the information to.

SaveFileType: An optional parameter saying the format of the file to be saved. If none is specified, then PWB will be assumed. My be one of the following strings:

PWB, PWB5, PWB6, PWB7. PWB8, PWB9, PWB10, PWB11, PWB12, PWB13
PTI (means PTI30) PTI23, PTI24, PTI25, PTI26, PTI27, PTI28, PTI29, PTI30
GE (means GE15), GE14, GE15
CF
AUX

WithTies: An optional parameter. One must specify the file type explicitly in order to use the WithTies parameter. Allows one to save transmission lines that tie the a bus marked with BusEquiv as false and one marked true. This must be a string which starts with the letter Y, otherwise NO will be assumed.

Scale(scaletype, basedon, [parameters], scalemarker);

Use this action to scale the load and generation in the system.

scaletype: The objecttype begin scaled. Must be either LOAD, GEN, INJECTIONGROUP, or BUSSHUNT.

basedon: MW: parameters are given in MW, MVAR units.

FACTOR: parameters a factor to multiple the present values by.

[parameters]: These parameters have different meanings depending on ScaleType.

LOAD: [MW, MVAR] or [MW]. If you want to scale load using constant power factor, then do not specifying a MVAR value.

GEN: [MW].

INJECTIONGROUP: [MW, MVAR] or [MW] . If you want to scale load using constant power factor, then do not specifying a MVAR value.

BUSSHUNT: [GMW, BCAPMVAR, BREAMVAR]. The first values scales G shunt values, the second value scales positive (capacitive) B shunt values, and the third value scales negative (reactive) B shunt values.

scalermarker: This value specifies whether to look at an element's bus, area, or zone to determine whether it should be scaled.

BUS: Means that elements will be scaled according to the BusScale property of the element's terminal bus.

AREA: Means that elements will be scaled according to the BGScale property of the element's Area. Note that it is possible for the area of a load, generator, or switched shunt to be different than the terminal bus's area.

ZONE: Means that elements will be scaled according to the BGScale property of the element's Zone. Note that it is possible for the zone of a load, generator, or switched shunt to be different than the terminal bus's zone.

OWNER: Means that the elements will be scaled according to the BGScale property of the element's Owner.

Move([elementA], [destination parameters]);

Use this action to move a generator, load, or transmission line.

[elementA]: the object that should be moved. Must be one of the following formats.

[GEN busnum id], [GEN "name_nomkv" id], [GEN "buslabel" id], [GEN "label"]

[LOAD busnum id], [LOAD "name_nomkv" id], [LOAD "buslabel" id], [LOAD "label"]

[BRANCH busnum1 busnum2 ckt], [BRANCH "name_kv1" "name_kv2" ckt],

[BRANCH "buslabel1" "buslabel2" ckt], [BRANCH "label"]

[destination parameters]: These parameters have different meanings depending on object type of the element. Bus numbers must be used here.

GEN: [busnum id]

LOAD: [busnum id]

BRANCH: [busnum1 busnum2 id]

Combine([elementA], [elementB]);

NOTE: THIS ACTION IS ONLY AVAILABLE FOR GENERATORS.

Use this action to combine two generators, two loads, or two transmission line. Note that elementA and elementB must be of the same object type. You can not combine a BRANCH and a LOAD.

[elementA]: The object that should be moved. Must be one of the following formats.

[GEN busnum id]

[LOAD busnum id]

[BRANCH busnum1 busnum2 ckt]

[elementB]: The object that element A should be combined with. Must the same format as for elementA.

SplitBus([element], NewBusNumber, InsertBusTieLine, LineOpen);

Use this action to transform one bus into two connected buses.

[element]: The bus that should be split. Must be of the following format:

[BUS num]

[BUS "name_nomkv"]

[BUS "buslabel"]

NewBusNumber: The number of the new bus resulting from the split. Must be unique, that is, it cannot be a number that identifies another number in the case.

InsertBusTieLine Set to YES or NO. YES means to insert a very low-impedance bus tie between the bus to split and its offspring. The new branch will have an impedance of 0.0000 + j0.0001 ohms. NO means that no bus tie will be inserted. If this parameter is not specified, then YES is assumed.

LineOpen Set to YES or NO. YES means that the bus tie should be inserted as an open branch. NO means that the bus tie will be inserted as a closed branch. This options is valid only if InsertBusTieLine is set to YES. If this parameter is not specified, then NO is assumed.

MergeBuses([element], filter);

Use this action to merge a set of buses into a single bus.

[element]: The number of the new bus resulting from the merging. It should be a number of a bus that is part of the set of buses to be merged, or it should be unique, meaning that it cannot be a number that identifies another number in the case. Must be of the following format:

[BUS num]

filter: There are four options for the filter:

MergeBuses(...);: No filter specified means to merge all the buses of the system.

MergeBuses(..., "filtername");: "filtername" means to merge those buses that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

MergeBuses(..., AREAZONE);: AREAZONE means to merge those buses that meet the area/zone filters.

MergeBuses(..., SELECTED);: SELECTED means to merge those buses whose *selected* field is set to YES.

TapTransmissionLine([element], PosAlongTheLine, NewBusNumber, ShuntModel, TreatAsMSLine);

Use this action to insert a bus at some location along an existing transmission line.

[element]: The transmission line to tap. Must be of the following format:

[BRANCH busnum1 busnum2 ckt]

[BRANCH "name_kv1" "name_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

PosAlongTheLine: The value that indicates the point where the new bus is to be inserted relative to the location of the nearest of the two buses in terms of a percentage of the total line length This value controls how the impedances of the new lines are set, as the impedance of each section will equal the section's corresponding percentage length multiplies by the impedance of the original line.

NewBusNumber: The number of the new bus resulting from the tapping. Must be unique, that is, it cannot be a number that identifies another number in the case.

ShuntModel: The way the original charging capacitance of the line is reassigned. The choices are:

LINESHUNTS: The original charging capacitance is reassigned as line shunts at the original terminal bus ends of the two new line segments. The charging capacitance of the two new branch elements will be set to 0.

CAPACITANCE: The original charging capacitance of the line is converted approximately using percentage entered as PosAlongTheLine. In other words, the original capacitance will be scaled and assigned to the new segments as charging capacitance.

The default ShuntModel is LINESHUNTS.

TreatAsMSLine: Set to YES or NO. YES means that a new multi-section line will be created that is comprised of the two new lines. NO means that the two new lines will be controlled independently of each other.

InterfacesAutoInsert(Type, DeleteExisting, UseDisplayFilters, Prefix, Limits);

Use this action to automatically insert a group of interfaces. The new of the new interfaces defaults to "Area1-Area2" or "Zone1-Zone2" with an optional **Prefix**.

Type: The type of interfaces to insert. Choices are:

AREA: Area-to-area interfaces join adjacent areas (those that share at least one tie line).

ZONE: Zone-to-zone interfaces join adjacent interfaces (those that share at least one tie line).

DeleteExisting: Set to YES or NO. YES means that previously defined interfaces will be deleted before the new interfaces are automatically inserted. NO means automatically inserted interfaces will be added to the list of previously defined interfaces. If this parameter is not specified, then YES is assumed.

UseDisplayFilters: Set to YES or NO. YES means that only Areas and Zones with their Area/Zone filters set to YES will be used when automatically inserting interfaces. NO specifies all Areas and Zones will be used to automatically insert interfaces regardless of their Area/Zone filters. If this parameter is not specified, then NO is assumed.

Prefix: An optional prefix of up to three characters. Use this prefix to avoid duplicating names, particularly when some of the areas or zones have the same name. If this parameter is not specified, then "" is assumed.

Limits: Specifies the way the interfaces rating are set. Choices are:

AUTO: The interface rating is calculated based on the ratings of the components included in the interface.

ZEROS: All the interface limits are set to zero.

[value1, ..., value8]: The set of rating values to be used for the interfaces. The eight values must be specified, separated by commas.

The default parameter is ZEROS.

Script Run Mode Actions

The following script commands are available during any of the submodes of Run Mode:

Animate;

CalculatePTDF	([transactor seller],	[transactor buyer],	LinearMethod);
	[AREA num]	[AREA num]	AC
	[ZONE num]	[ZONE num]	DC
	[SUPERAREA "name"]	[SUPERAREA name]	DCPS
	[INJECTIONGROUP "name"]	[INJECTIONGROUP name]	
	[BUS num]	[BUS num]	
	[SLACK]	[SLACK]	
CalculatePTDFMultipleDirections	(StoreValuesForBranches,	StoreValuesForInterfaces,	LinearMethod);
	YES or NO	YES or NO	AC
			DC
			DCPS

CalculateLODF	([BRANCH nearbusnum farbusnum ckt],	LinearMethod);
---------------	-------------------------------------	----------------

CalculateLODFMatrix	(WhichOnes,	filterProcess,	filterMonitor,	MonitorOnly Closed,	Linear Method);
	OUTAGES	ALL	ALL	YES	DC
	CLOSURES	SELECTED	SELECTED	NO	DCPS
		AREAZONE "filtername"	AREAZONE "filtername"		
			SAME		

CalculateTLR	([flowelement],	direction,	[transactor],	LinearMethod);
	[INTERFACE "name"]	BUYER	same as above for PTDFs	
	[BRANCH nearbusnum farbusnum ckt]	SELLER		

CalculateTLRMultipleElement	(TypeElement	WhichElement,	direction,	[transactor],	Linear Method);
	BRANCH	SELECTED	BUYER	same as above for PTFDs	
	INTERFACE	OVERLOAD	SELLER		
	BOTH	CTGOVERLOAD			
CalculateVoltSense	([BUS num]);				
CalculateFlowSense	([flowelement],	FlowType);			
	[INTERFACE "name"]	MW			
	[BRANCH busnum1 busnum2 ckt]	MVAR MVA			
CalculateLossSense	(FunctionType);				
	NONE				
	ISLAND				
	AREA				
	AREASA				
	SELECTED				
CalculateVoltToTransferSense	([transactor seller],	[transactor buyer],	TransferType	TurnOffAVR);	
	[AREA num]	[AREA num]	[P, Q, or PQ]	YES or NO	
	[ZONE num]	[ZONE num]			
	[SUPERAREA "name"]	[SUPERAREA name]			
	[INJECTIONGROUP "name"]	[INJECTIONGROUP name]			
	[BUS num]	[BUS num]			
	[SLACK]	[SLACK]			
CalculateVoltSelfSense	(filter);				

SetSensitivitiesAtOutOfServiceToClosest;

ZeroOutMismatches;

Animate(DoAnimate);

Use this action to animate all the open online diagrams.

DoAnimate: Set to YES or NO. YES means to start the animation of the open online diagrams, while NO means that the animation will be paused.

CalculatePTDF([transactor seller], [transactor buyer], LinearMethod);

Use this action to calculate the PTDF values between a seller and a buyer. You may optionally specify the linear calculation method. Note that the buyer and seller must not be same thing. If no Linear Method is specified, Lossless DC will be used.

[transactor seller]: The seller (or source) of power. There are six possible settings:

[AREA num], [AREA "name"], [AREA "label"]
 [ZONE num], [ZONE "name"], [ZONE "label"]
 [SUPERAREA "name"], [SUPERAREA "label"]
 [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
 [BUS num], [BUS "name_nomkv"], [BUS "label"]
 [SLACK]

[transactor buyer]: The buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

LinearMethod: The linear method to be used for the PTDF calculation. The options are:

AC: Calculation including losses.
 DC: Lossless DC.
 DCPS: Lossless DC that takes into account phase shifter operation.

CalculatePTDFMultipleDirections(StoreValuesForBranches, StoreValuesForInterfaces, LinearMethod);

Use this action to calculate the PTDF values using multiple directions. Only directions with the field INCLUDE set to YES will be analyzed when calculating the multiple direction PTDFs. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

StoreValuesForBranches: Set to YES or NO. YES means that the PTDFs for lines/transformers will be stored.

StoreValuesForInterfaces: Set to YES or NO. YES means that the PTDFs for interfaces will be stored.

LinearMethod: The linear method to be used for the PTDF calculation. The options are:

AC: Calculation including losses.
 DC: Lossless DC.
 DCPS: Lossless DC that takes into account phase shifter operation.

CalculateLODF([BRANCH nearbusnum farbusnum ckt], LinearMethod);

Use this action to calculate the Line Outage Distribution Factors (or the Line Closure Distribution Factors) for a particular branch. If the branch is presently closed, then the LODF values will be calculated, otherwise the LCDF values will be calculated. You may optionally specify the linear calculation method as well. If no Linear Method is specified, Lossless DC will be used.

[BRANCH nearbusnum farbusnum ckt]: The branch whose status is being changed. Can also use strings

[BRANCH "nearbusname_kv" "farbusname_kv" ckt]

[BRANCH "nearbuslabel" "farbuslabel" ckt]

[BRANCH "label"]

LinearMethod: The linear method to be used for the LODF calculation. The options are:

DC: Lossless DC.

DCPS: Lossless DC that takes into account phase shifter operation.

Note: AC is NOT an option for the LODF calculation.

CalculateLODFMatrix(WhichOnes, filterProcess, filterMonitor, MonitorOnlyClosed, LinearMethod);

Use this action to calculate a matrix of LODF results comprised of multiple branches to be outaged or closed and multiple branches for which the impact of the outages/closures is determined.

WhichOnes: Specify the type of sensitivities to be calculated.

OUTAGES: Outage sensitivities will be calculated for those branches meeting the filterProcess.

CLOSURES: Closure sensitivities will be calculated for those branches meeting the filterProcess.

filterProcess: Specify a filter for the branches for which the outages or closures will be implemented.

ALL: All AC transmission lines.

SELECTED: Only those branches whose Selected field is set to *YES*.

AREAZONE: Only those branches meeting the area/zone filter.

"filename": Name of advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filename" then takes on the form "<objecttype>objecttype's filename".

filterMonitor: Specify a filter for the branches for which the impact of the outages or closures will be determined.

ALL: All AC transmission lines.

SELECTED: Only those branches whose Selected field is set to *YES*.

AREAZONE: Only those branches meeting the area/zone filter.

"filename": Name of advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filename" then takes on the form "<objecttype>objecttype's filename".

SAME: Same as set of branches to process as specified by filterProcess.

MonitorOnlyClosed: Set to YES to monitor only those branches that are closed. Set to NO to monitor branches regardless of their status.

LinearMethod: The linear method to use in the LODF calculations.

DC: Lossless DC.

DCPS: Lossless DC that takes into account phase shifter operation.

CalculateTLR([flow element], direction, [transactor], LinearMethod);

Use this action to calculate the TLR values a particular flow element (transmission line or interface). You also specify one end of the potential transfer direction. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

[flow element]: This is the flow element we are interested in. Choices are:

- [INTERFACE "name"]
- [INTERFACE "label"]
- [BRANCH nearbusnum farbusnum ckt]
- [BRANCH "nearbusname_kv" "farbusname_kv" ckt]
- [BRANCH "nearbuslabel" "farbuslabel" ckt]
- [BRANCH "label"]

direction: the type of the transactor. Either BUYER or SELLER.

[transactor buyer]: the transactor of power. There are six possible settings.

- [AREA num], [AREA "name"], [AREA "label"]
- [ZONE num], [ZONE "name"], [ZONE "label"]
- [SUPERAREA "name"], [SUPERAREA "label"]
- [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
- [BUS num], [BUS "name_nomkv"], [BUS "label"]
- [SLACK]

LinearMethod: The linear method to be used for the calculation. The options are:

- AC: Calculation including losses.
- DC: Lossless DC.
- DCPS: Lossless DC that takes into account phase shifter operation.

CalculateTLRMultipleElement(TypeElement, WhichElement, direction, [transactor], LinearMethod);

Use this action to calculate the TLR values a multiple elements similar to as is done on the TLR multiple elements dialog. You also specify one end of the potential transfer direction. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

TypeElement : May be either INTERFACE, BRANCH, or BOTH

WhichElement : There are three choices which represent which elements of the TypeElement specified will have TLR calculations performed.

- SELECTED : Only branches or interfaces with their Selected Field = YES will be used.
- OVERLOAD : Only branches that are presently overloaded using their normal ratings will be used
- CTGOVERLOAD : You must have first run the contingency analysis. A branch or interface is included in the calculation if it has been overloaded during at least one contingency.

direction: the type of the transactor. Either BUYER or SELLER.

[transactor buyer]: the transactor of power. There are six possible settings.

[AREA num], [AREA "name"], [AREA "label"]
 [ZONE num], [ZONE "name"], [ZONE "label"]
 [SUPERAREA "name"], [SUPERAREA "label"]
 [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
 [BUS num], [BUS "name_nomkv"], [BUS "label"]
 [SLACK]

LinearMethod: The linear method to be used for the calculation. The options are:

AC: Calculation including losses.

DC: Lossless DC.

DCPS: Lossless DC that takes into account phase shifter operation.

CalculateVoltSense([BUS num]);

This calculates the sensitivity of a particular buses voltage to real and reactive power injections at all buses in the system. (Note: this assumes that the power is injected at a given bus and taken out at the slack bus).

[BUS num]: The bus to calculate sensitivities for.

CalculateFlowSense([flow element], FlowType);

This calculates the sensitivity of the MW, MVAR, or MVA flow of a line or interface to real and reactive power injections at all buses in the system. (Note: this assumes that the power is injected at a given bus and taken out at the slack bus).

[flow element]: This is the flow element we are interested in. Choices are:

[INTERFACE "name"]
 [INTERFACE "label"]
 [BRANCH busnum1 busnum2 ckt]
 [BRANCH "name_kv1" "name_kv2" ckt]
 [BRANCH "buslabel1" "buslabel2" ckt]
 [BRANCH "label"]

FlowType: The type of flow to calculate this for. Either MW, MVAR, or MVA.

CalculateLossSense(FunctionType);

This action calculates the sensitivity of a real power loss function, PLosses, to bus real and reactive power injections. Stated mathematically, this action calculates $d PLosses/d P_i$ and $d PLosses/d Q_i$, where P_i and Q_i are the real and reactive power injections at bus i , respectively. Stated less formally, this action indicates how losses would change if one more MW or Mvar of power were injected at bus i .

FunctionType: Specifies how the losses are computed. Choices are:

NONE: No losses are calculated because a loss function is not specified.

ISLAND: Losses are calculated with respect to the losses in bus' island. If the power system consists of only one island, losses are computed with respect to the total system losses.

AREA: Losses are calculated with respect to the total losses for the area containing bus i. This is probably the most common loss function because usually one is concerned with minimizing losses for a particular area rather than for the entire case.

AREASA: Losses are calculated with respect to the total losses for the area containing bus i if bus i does not belong to a super area, and with respect to the total losses for the super area containing bus i if bus i does belong to a super area.

SELECTED: Losses are calculated with respect to the total losses for a group of areas whose *selected* field is set to YES.

CalculateVoltToTransferSense([transactor seller], [transactor buyer], TransferType, TurnOffAVR);

This calculates the sensitivity of bus voltage to a real or reactive power transfer between a seller and a buyer. The sensitivity is calculated for all buses in the system.

[transactor seller]: This is the seller (or source) of power. There are six possible settings:

- [AREA num], [AREA "name"], [AREA "label"]
- [ZONE num], [ZONE "name"], [ZONE "label"]
- [SUPERAREA "name"], [SUPERAREA "label"]
- [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
- [BUS num], [BUS "name_nomkv"], [BUS "label"]
- [SLACK]

[transactor buyer]: This is the buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

TransferType: The type of power transfer. The options are:

- P – real power transfer
- Q – reactive power transfer
- PQ – both real and reactive power transfer. (Note: Real and reactive power transfers are calculated independently, but both are calculated.)

TurnOffAVR: Set to YES or NO. Set to YES to turn off AVR control for generators participating in the transfer. Set to NO to leave the AVR control unchanged for generators participating in the transfer.

CalculateVoltSelfSense(filter);

This calculates the sensitivity of a particular bus' voltage to real and reactive power injections at the same bus. (Note: This assumes that the power is injected at a given bus and taken out at the slack bus.)

filter: There are four options for the filter:

CalculateVoltSelfSense; : No filter specified means to merge all the buses of the system.

CalculateVoltSelfSense("filtername"); : "filtername" means to merge those buses that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

CalculateVoltSelfSense(AREAZONE); : AREAZONE means to merge those buses that meet the area/zone filters.

CalculateVoltSelfSense(SELECTED); : SELECTED means to merge those buses whose *selected* field is set to YES.

SetSensitivitiesAtOutOfServiceToClosest;

This will take the P Sensitivity and Q Sensitivity values calculated using CalculateTLR, CalculateFlowSense, or CalculateVoltSense actions and then populate the respective values at out-of-service buses so that they are equal to the value at the closest in service bus. The "distance" to the in-service buses will be measured by the number of nodes. If an out-of-service bus is equally close to a set of buses, then the average of that set of buses will be used.

ZeroOutMismatches;

With this script command, the bus shunts are changed at each bus that has a mismatch greater than the MVA convergence tolerance so that the mismatch at that bus is forced to zero.

In addition to the general run mode actions, there are specific actions for the following submodes: [PowerFlow](#), [Contingency](#), [ATC](#), [Fault](#), [PV](#) and [QV](#).

Script PowerFlow Submode Actions

The following actions are specific to the Run Mode submode PowerFlow:

```
DoCTGAction([contingency action]);
SolvePowerFlow(SolMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
ResetToFlatStart(FlatVoltagesAngles, ShuntsToMax, LTCsToMiddle, PSAnglesToMiddle);
SolvePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
InitializePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
SolveSinglePrimalLPOuterLoop("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
SolveFullSCOPF(BCMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
OPFWriteResultsAndOptions ("filename");
DiffFlowSetAsBase;
DiffFlowClearBase;
DiffFlowMode (diffmode);
PRESENT
BASE
DIFFERENCE
```

DoCTGAction([contingency action]);

Call this action to use the formats seen in the CTGElement subdata record for Contingency Data. Note that all actions are supported, except COMPENSATION sections are not allowed.

SolvePowerFlow(SolMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

Call this action to perform a single power flow solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

SolMethod The solution method to be used for the power flow calculation. The options are:

RECTNEWT: For Rectangular Newton-Raphson method.

POLARNEWTON: For Polar Newton-Raphson method.

GAUSSSEIDEL: For Gauss-Seidel method.

FASTDEC: For Fast Decoupled method.

DC: For DC method.

ROBUST: For ROBUST method.

The default method is RECTNEWT.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

ResetToFlatStart (FlatVoltagesAngles, ShuntsToMax, LTCsToMiddle, PSAnglesToMiddle);

Use this action to initialize the Power Flow Solution to a "flat start." The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

FlatVoltagesAngles Set to YES or NO. YES means setting all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero. Default Value is YES.

ShuntsToMax Set to YES or NO. YES means to increase Switched Shunts Mvar half way to maximum. Default Value is NO.

LTCsToMiddle Set to YES or NO. YES means setting the LTC Transformer Taps to middle of range. Default Value is NO.

PSAnglesToMiddle Set to YES or NO. YES means setting Phase Shifter angles to middle of range. Default Value is NO.

SolvePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

Call this action to perform a primal LP OPF solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

SolveSinglePrimalLPOuterLoop("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

This action is basically identical to the SolvePrimalLP action, except that this will only perform a single optimization. The SolvePrimalLP will iterate between solving the power flow and an optimization until this iteration converges. This action will only solve the optimization routine once, then resolve the power flow once and then stop.

InitializePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

This command clears all the structures and results of previous primal LP OPF solutions. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

SolveFullSCOPF (BCMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

Call this action to perform a full Security Constrained OPF solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

BCMethod The solution method to be used for solving the base case. The options are:

POWERFLOW – for single power flow algorithm.

OPF – for the optimal power flow algorithm.

Default Value is POWERFLOW.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

OPFWriteResultsAndOptions("filename");

Writes out all information related to OPF analysis as an auxiliary file. This includes Limit Monitoring Settings, options for Areas, Buses, Branches, Interfaces, Generators, SuperAreas, OPF Solution Options.

DiffFlowSetAsBase;

Call this action to set the present case as the base case for the difference flows abilities of Simulator.

DiffFlowClearBase;

Call this action to clear the base case for the difference flows abilities of Simulator.

DiffFlowMode(diffmode);

Call this action to change the mode for the difference flows abilities of Simulator.

diffmode:

String that starts with 'P' changes it to PRESENT

String that starts with 'B' changes it to BASE

String that starts with 'D' changes it to DIFFERENCE

Script Contingency Submode Actions

The following actions are specific to the Run Mode Submode Contingency:

```
CTGSolveAll;
CTGSolve("ContingencyName");
CTGSetAsReference;
CTGRestoreReference;
CTGProduceReport("filename");
CTGWriteResultsAndOptions("filename");
CTGAutoInsert;
CTGCalculateOTDF( [transactor seller], [transactor buyer], LinearMethod);
[AREA num] [AREA num] AC
[ZONE num] [ZONE num] DC
[SUPERAREA "name"] [SUPERAREA "name"] DCPS
[INJECTIONGROUP "name"] [INJECTIONGROUP "name"]
[BUS num] [BUS num]
[SLACK] [SLACK]
```

CTGSolveAll;

Call this action to solve all the contingencies which are not marked skip.

CTGSolve("ContingencyName");

Call this action solve a particular contingency. The contingency is denoted by the "Contingency Name".

CTGSetAsReference;

Call this action to set the present system state as the reference for contingency analysis.

CTGRestoreReference;

Call this action to reset the system state to the reference state for contingency analysis.

CTGProduceReport("filename");

Produces a text-based contingency analysis report using the settings defined in CTG_Options.

CTGWriteResultsAndOptions("filename");

Writes out all information related to contingency analysis as an auxiliary file. This includes Contingency Definitions, Limit Monitoring Settings, Contingency Results, Solution Options, CTG Options as well as any Model Criteria that are used by the Contingency Definitions.

CTGAutoInsert;

Automatically generates a contingency list using the settings defined in CTG_Options.

CTGCalculateOTDF([transactor seller], [transactor buyer], LinearMethod);

This action calculate OTDFs for each contingency-caused branch or interface violation. The values calculated will be a measure of what percent of a transfer would appear on the branch or interface after the respective contingency occurs. Realize for branch violations, that the sign of the OTDF value will be relative to the direction of the MW flow found during the contingency analysis.

The parameters will be used to calculate the PTDFs for the transfer direction you are interested in. These PTDF values will then be used throughout the OTDF calculation. Note that the buyer and seller must not be same thing. If no Linear Method is specified, Lossless DC will be used.

NOTE: All the parameters are optional. If they are not specified, then the existing PTDFs will be used.

[transactor seller]: The seller (or source) of power. There are six possible settings:

[AREA num]

[ZONE num]

[SUPERAREA "name"]

[INJECTIONGROUP "name"]

[BUS num]

[SLACK]

[transactor buyer]: The buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

LinearMethod: The linear method to be used for the PTDF calculation. The options are:

AC: for calculation including losses.

DC: for lossless DC.

DCPS: for lossless DC that takes into account phase shifter operation.

Script ATC Submode Actions

The following actions are specific to the Run Mode Submode ATC:

ATCDetermine	([transactor seller],	[transactor		
	[AREA num]	buyer]);		
	[ZONE num]	[AREA num]		
	[SUPERAREA	[ZONE num]		
	"name"]	[SUPERAREA		
		name]		
	[INJECTIONGROUP	INJECTIONGROUP		
	"name"]	name]		
	[BUS num]	[BUS num]		
	[SLACK]	SLACK]		
ATCRestoreInitialState;				
ATCIncreaseTransferBy	(amount);			
ATCTakeMeToScenario	(RL,	G,	I);	
ATCDetermineATCFor	(RL,	G,	I,	ApplyTransfer);
ATCWriteResultsAndOptions	("filename",	AppendFile);		
ATCWriteToExcel	("worksheetname");			
ATCWriteToText	("filename",	filetype);		

ATCDetermine([transactor seller], [transactor buyer]);

Use this action to calculate the Available Transfer Capability (ATC) between a seller and a buyer. Note that the buyer and seller must not be same thing. Other options regarding ATC calculations should be set using a DATA section via the ATC_Options object type.

[transactor seller]: The seller (or source) of power. There are six possible settings

[AREA num], [AREA "name"], [AREA "label"]
 [ZONE num], [ZONE "name"], [ZONE "label"]
 [SUPERAREA "name"], [SUPERAREA "label"]
 [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
 [BUS num], [BUS "name_nomkv"], [BUS "label"]
 [SLACK]

[transactor buyer]: The buyer (or sink) of power. There are six possible settings which are the same as for the seller.

ATCRestoreInitialState;

Call this action to restore the initial state for the ATC tool.

ATCIncreaseTransferBy(amount);

Call this action to increase the transfer between the buyer and seller.

ATCTakeMeToScenario(RL, G, I);

Call this action to set the present case according to Scenario RL, G, I.

ATCDetermineATCFor(RL, G, I, ApplyTransfer);

Call this action to determine the ATC for Scenario RL, G, I.

ApplyTransfer: Set this value to YES to leave the system state at the transfer level that was determined. When using the Iterated Linear then Full Contingency solution method, the system state will retain the transfer level but the contingency will not be applied.

ATCWriteResultsAndOptions("filename", AppendFile);

Writes out all information related to ATC analysis as an auxiliary file. This includes Contingency Definitions, Limit Monitoring Settings, Solution Options, DataGrid settings relevant to ATC, ATC Options, ATC results, as well as any Model Criteria that are used by the Contingency Definitions.

"filename": Name of the auxiliary file.

AppendFile: This is an optional parameter. YES means to append results to existing "filename". NO means to overwrite "filename" with results. Default is YES.

ATCWriteToExcel("worksheetname");

Sends ATC analysis results to an Excel spreadsheet. This script command is available only for Multiple Scenarios ATC analysis.

"worksheetname": The name of the Excel sheet where the results will be sent.

ATCWriteToText("filename", filetype);

This is used with Multiple Scenario ATC analysis. Multiple files are created with "filename" as the primary identifier and the Interface scenario label appended to the end of the filename. Separate files are created for each of the Interface scenarios. Results inside the files are separated into sections based on the number of Rating/Load scenarios.

"filename": Primary identifier for the name of the file in which to save the results. "filename" gets appended with the Interface scenario label to complete the filename.

filetype: Either TAB or CSV. This indicates the delimiter to use when writing out the file(s). This is an optional parameter with TAB being the default if omitted.

Script Fault Submode Actions

The following actions are specific to the Run Mode Submode Fault:

```

Fault      ([BUS
            num],
            faulttype,   R,   X);
Fault      ([BRANCH    faultlocation,   faulttype,   R,   X);
            nearbusnum
            farbusnum
            ckt],
            SLG
            LL
            3PB
            DLG

```

Fault([Bus num, faulttype, R, X]);

Fault([BRANCH nearbusnum farbusnum ckt], faultlocation, faulttype, R, X);

Call this function to calculate the fault currents for a fault. If the fault element is a bus then do not specify the fault location parameter. If the fault element is a branch, then the fault location is required.

[Bus num]: This specifies the bus at which the fault occurs. You may also specify the bus using secondary keys or labels:

[BUS "name_nomkv"]

[BUS "label"]

[BRANCH nearbusnum farbusnum ckt]: This specifies the branch on which the fault occurs. You may also specify the branch using secondary keys or labels:

[BRANCH "name_kv1" "name_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

Fault location: This specifies the percentage distance along the branch where the fault occurs. This percent varies from 0 (meaning at the nearbus) to 100 (meaning at the far bus)

Faulttype: This specifies the type of fault which occurs. There are four options:

SLG: Single Line To Ground fault

LL: Line to Line Fault

3PB: Three Phase Balanced Fault

DLG: Double Line to Group Fault.

R, X: These parameters are optional and specify the fault impedance. If none are specified, then a fault impedance of zero is assumed.

Script PV Submode Actions

The following actions are specific to the Run Mode Submode PV:

```

PVSetSourceAndSink      ([elementSource],      [elementSink]);
                        [INJECTIONGROUP      [INJECTIONGROUP
                        "name"]              "name"]

PVRun;

PVRun      ([elementSource],      [elementSink]);
            [INJECTIONGROUP "name"] [INJECTIONGROUP "name"]

PVClear;
PVStartOver;
PVDestroy;

PVWriteResultsAndOptions ("filename");

RefineModel (objecttype, filter, Action      ForWhichValue);
            AREA          TRANSFORMERTAPS
            ZONE          SHUNTS
                        OFFFAVR
    
```

Changes were made with Simulator version 14 to eliminate the need for a PV study name. To maintain functionality with any existing processes that users might have in place using older script definitions, scripts from older versions of Simulator will still be supported if the name is specified. However, the name will just be ignored. The script formats given here reflect the changes for version 14.

The PVCreate script required in previous versions is no longer necessary in Simulator version 14. Version 14 will still recognize this action if it is included and will simply set the source and sink for the study. This does the same thing as PVSetSourceAndSink.

It is highly recommended that for any new processes the new script formats specified here be used.

PVSetSourceAndSink([elementSource], [elementSink]);

Call the function to specify the source and sink elements to perform the PV study.

[element source]: The source of power for the PV study. There is only one possible setting:
 [INJECTIONGROUP "name"] or [INJECTIONGROUP "label"]

[element sink]: The sink of power for the PV study. There is only one possible setting, which is the same as for the source.

PVRun;

Call the function to start the PV study.

PVRun([elementSource], [elementSink]);

Call the function to specify the source and sink elements and start the PV study.

[element source]: The source of power for the PV study. There is only one possible setting:

[INJECTIONGROUP "name"] or [INJECTIONGROUP "label"]

[element sink]: The sink of power for the PV study. There is only one possible setting, which is the same as for the source.

PVClear;

Call the function to clear all the results of the PV study.

PVStartOver;

Call the function to start over the PV study. This includes clear the activity log, clear results, restore the initial state, set the current state as initial state, and initialize the step size.

PVDestroy;

Call the function to destroy the PV study. This will remove all results and prevent any restoration of the initial state that is stored with the PV study.

PVWriteResultsAndOptions("filename");

Call this action to save all the PV results and options in the auxiliary file "filename".

RefineModel(objecttype, filter, Action, Tolerance);

Call this function to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during PV and QV studies.

objecttype: The objecttype being selected.

AREA

ZONE

filter: There are three options for the filter:

RefineModel(..., "", ...);: No filter specified means to select all objects of this type.

RefineModel(..., "filtername", ...);: "filtername" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

Action: The way the model will be refined. Choices are:

TRANSFORMERTAPS Fix all transformer taps at their present values if their $V_{max} - V_{min}$ is less than or equal to the user specified tolerance.

SHUNTS Fix all shunts at their present values if their $V_{max} - V_{min}$ is less than or equal to the user specified tolerance.

OFFAVR Remove units from AVR control, thus locking their MVAR output at its present value if their $Q_{max} - Q_{min}$ is less or equal to the user specified tolerance.

Tolerance: Tolerance value.

Script QV Submode Actions

The following actions are specific to the Run Mode Submode QV:

```
QVRun      ("filename",      InErrorMakeBaseSolvable);
           YES or NO
```

NOTE: The QV study is always performed on selected buses.

```
QVWriteResult  ("filename");
sAndOptions
```

```
RefineModel  (objecttype,      filter,      Action      ForWhichValue);
              AREA
              ZONE
              TRANSFORMERTAPS
              SHUNTS
              OFFAVR
```

QVRun("filename", InErrorMakeBaseSolvable);

Call the function to start a QV study for the list of buses whose SELECTED? field is set to YES.

"filename": This specifies the file to which to save a comma-delimited version of the results.

InErrorMakeBaseSolvable: This specifies whether to perform a solvability analysis of the base case if the pre-contingency base case can not be solved. If not specified, then YES is assumed.

QVWriteResultsAndOptions("filename");

Call this action to save all the QV results and options in the auxiliary file "filename".

RefineModel(objecttype, filter, Action, Tolerance);

Call this function to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during PV and QV studies.

objecttype: The objecttype being selected.

AREA

ZONE

filter: There are three options for the filter:

RefineModel(..., "", ...);: No filter specified means to select all objects of this type.

RefineModel(..., "filtername", ...);: "filtername" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

Action: The way the model will be refined. Choices are:

TRANSFORMERTAPS Fix all transformer taps at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

SHUNTS Fix all shunts at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

OFFAVR Remove units from AVR control, thus locking their MVAR output at its present value if their $Q_{max} - Q_{min}$ is less or equal to the user specified tolerance.

Tolerance: Tolerance value.

Data Section

The Data Section of the script file format can contain numerous functions for modifying the data of a power system case. The DATA section begins with a left curly brace and ends with a right curly brace.

DATA DataName (object_type, [list_of_fields], file_type_specifier)

```
{  
data_list_1  
.  
.  
.  
data_list_n  
}
```

Immediately following the DATA keyword, you may optionally include a DataName. By including the DataName, you can make use of the script command LoadData("filename", DataName) to call this particular data section from another auxiliary file.

The first line of the DATA section is referred to as the [Data Argument List](#). The [Data Argument List](#) defines the type and format of the data contained in the following section enclosed in curly braces.

Inside of each DATA section, [data lists](#) are given according to the data definition given in the [Data Argument List](#).

Data Argument List

The DATA argument list identifies what the information section contains. A left and right parenthesis "(" and ")" mark the beginning and end of the argument list.

The **file_type_specifier** parameter distinguishes the information section as containing custom auxiliary data (as opposed to Simulator's native auxiliary formats), and indicates the format of the data. Currently, the parser recognizes two values for **file_type_specifier**:

(blank) or AUXDEF	Data fields are space delimited
CSV, AUXCSV, or CSVAUX	Data fields are comma delimited

The **create_if_not_found** parameter allows the specification of whether or not one or more data records that exist in the data section that do not link to existing data in the case be created when read from the auxiliary file. The parameter is optional, and if omitted Simulator will prompt the user regarding creating objects not found in the case from records in the data section. If the parameter is specified as True, then objects not found will be created from the data records, and if False objects not found will not be created from the data records. Data records for which objects already exist in the model will be read and applied regardless of this parameter setting.

The **object_type** parameter identifies the type of object or data element the information section describes or models. For example, if **object_type** equals BUS, then the data describes BUS objects. Below is a list of many of the object types recognized by Simulator:

3WXFormer	DefDrawSuperArea	OPF_Options_and_Results
Area	DefDrawZone	OPFSolutionSummary
ATC_Options	Direction	Owner
ATCExtraMonitor	DynamicFormatting	PieSizeColorOptions
ATCGeneratorChange	Equiv_Options	PostPowerFlowActions
ATCInterfaceChange	ExportBidCurve	PTDF_Options
ATCLineChange	Filter	PVCurve_Options
ATCLineChangeB	GEMotor	PWCaseInformation
ATCScenario	Gen	PWFormOptions
ATCZoneChange	GeoDataViewStyle	PWLPOPFGTGViol
AuxFileExportFormatData	GlobalContingencyActions	PWLPTabRow
AuxFileExportFormatDisplay	GScenarioName	PWOPFTimePoint
BGCalculatedField	HintDefValues	PWPVResultListContainer
Branch	IG_AutoInsert_Options	QVCurve
Bus	ImportBidCurve	QVCurve_Options
BusGroupSwapRec	ImportExportBidCurve	RLScenarioName
BusNumberSwap	InjectionGroup	Scale_Options
BusViewFormOptions	Interface	Schedule
CaseInfo_Options	InterfaceAElement	ScreenLayer
ColorMap	InterfaceBElement	SelectByCriteriaSet
Contingency	IScenarioName	ShapefileExportDescription
Ctg_AutoInsert_Options	Limit_Monitoring_Options	ShortCut

Ctg_Options	LimitCost	Shunt
CTGElementBlock	LimitSet	Sim_Environment_Options
CustomCaseInfo	Load	Sim_Misc_Options
CustomCaseInfoRow	LODF_Options	SimSimulation_Options
CustomColors	LPOPFMarginalControls	Sim_Solution_Options
CustomExpression	LPVariable	StudyMWTransactions
DataGrid	MessLog_Options	Substation
DCTransmissionLine	ModelCondition	SuperArea
DefDrawArea	ModelExpression	TLR_Options
DefDrawBackground	ModelFilter	TransferLimiter
DefDrawBus	MTDCRecord	Transformer
DefDrawGen	MultiSectionLine	TSSchedule
DefDrawInterface	MvarMarginalCostValues	UC_Options
DefDrawLineTransformer	MWMarginalCostValues	ViolationCTG
DefDrawLoad	MWTransaction	WhatOccurredDuringContingency
DefDrawShunt	Nomogram	XFCorrection
DefDrawSubstation	NomogramInterface	Zone

The list of object types Simulator’s auxiliary file parser can recognize will grow as new applications for the technology are found. Within Simulator, you will always be able to obtain a list of the available **object_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Object Fields**.

The **list_of_fields** parameter lists the types of values the ensuing records in the data section contain. The order in which the fields are listed in **list_of_fields** dictates the order in which the fields will be read from the file. Simulator currently recognizes over 3,000 different field types, each identified by a specific field name. Because the available fields for an object may grow as new applications are developed, you will always be able to obtain a list of the available **object_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Object Fields**. Certainly, only a subset of these fields would be found in a typical custom auxiliary file. In crafting applications to export custom auxiliary files, developers need concern themselves only the fields they need to communicate between their applications and Simulator. A few points of interest regarding the **list_of_fields** are:

- The **list_of_fields** may take up several lines of the text file.
- The **list_of_fields** should be encompassed by braces [].
- When encountering the PowerWorld comment string `'''` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'''` will be ignored as comments.
- Field names must be separated by commas.

Example DATA Argument List

```
DATA (BUS, [BusNomKV, Bus, // comment here
  ABCPhaseAngle: 1, ABCPhaseAngle: 2, ABCPhaseV, ABCPhaseV: 1,
  // comments allowed here to
```

```
// note that blank rows are ignored
AreaNum, BusAngle, BusB, BusCat, BusEquiv, BusG,
BusGenericSensV, BusKVolt, BusLambda, BusLoadMVA, // more comment
BusLoadMW, BusLongName],True)
```

One general note regarding the field names however. Some field names may be augmented with a field location. One example of this is the field LineMW. For a branch, there are two MW flows associated with the line: one MW flow at the from bus, and one MW flow at the to bus. So that the number of fields does not become huge, the same field name is used for both of these values. For the from bus flow, we write LineMW: 0, and for the to bus flow, we write LineMW: 1. Note that fieldnames such as LineMW: 0 may simply leave off the: 0.

The **create_if_not_found** field is optional. This specifies whether or not to create a new object if an existing one is not found. If the value is YES, objects will be created. If the value is NO, objects will not be created. If omitted, the default behavior of prompting the user about whether or not to create a non-existing object will continue. If loading an auxiliary file using the **LoadAux** script command, the **create_if_not_found** field for the data section will override the **CreateIfNotFound** field with the script.

Data Key Fields

Simulator uses certain fields to identify the specific object being described. These fields are called key fields. For example, the key field for BUS objects is BusNum, because a bus can be identified uniquely by its number. The key fields for GEN objects are BusNum and GenID. To properly identify each object, the object's key fields must be present. They can appear in any order in the **list_of_fields** (i.e. they need not be the first fields listed in **list_of_fields**). As long as the key fields are present, Simulator can identify the specific object. By going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Case Object Fields** you will obtain a list of fields available for each object type. In this output, the key fields will appear with asterisks *.

Data List

After the data argument list is completed, the Data list is given. The data section lists the values of the fields for each object in the order specified in **list_of_fields**. The data section begins with a left curly brace and ends with a right curly brace. A few points of interest regarding the **value_list**:

- The **value_list** may take up several lines of the text file.
- Each new data object must start on its own line of text.
- When encountering the PowerWorld comment string `'//'` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'//'` will be ignored as comments.
- Remember that the right curly brace must appear on its own line at the end of the **data_list**.
- If the **file_type_specifier** is CSV, the values should be separated by commas. Otherwise, separate the field names using spaces.
- Strings can be enclosed in double quotes, but this is not required. You should however always enclose strings that contain spaces (or commas) in quotes. Otherwise, strings containing commas would cause errors for comma-delimited files, and spaces would cause errors for space-delimited formatted files.

Data SubData Sections

The format described in the [Data Section](#) works well for most kinds of data in Simulator. It does not work as well however for data that stores a list of objects. For example, a contingency stores some information about itself (such as its name), and then a list of contingency elements, and possible a list of limit violations as well. For data such as this, Simulator allows `<subData>`, `</subData>` tags that store lists of information about a particular object. This formatting looks like the following:

DATA (object_type, [list_of_fields], file_type_specifier)

```
{
value_list_1
  <SUBDATA subobject_type1>
    precise format describing an object_type1
    precise format describing an object_type1
    .
    .
    .
  </SUBDATA>
  <SUBDATA subobject_type2>
    precise format describing an object_type2
    precise format describing an object_type2
    .
    .
    .
  </SUBDATA>
value_list_2
.
.
.
value_list_n
}
```

Note that the information contained inside the `<SubData>`, `</SubData>` tags may not be flexibly defined. It must be written in a precisely defined order that will be documented for each SubData type.

Data ATC_Options

RLScenarioName**GScenarioName****IScenarioName**

These three sections contain the pretty names of the RL Scenarios, G Scenarios, and I Scenarios. Each line consists of two values: Scenario Number and a name string enclosed in quotes.

Scenario Number: The scenarios are number 0 through the number of scenarios minus 1.

Scenario Name: These represent the names of the various scenarios.

Example:

```
<SUBDATA RLScenarioName>
//Index Name
0 "Scenario Name 0"
1 "Scenario Name 1"
</SUBDATA>
```

ATCMemo

This section contains the memo text for the ATC analysis.

Example:

```
<SUBDATA ATCMemo>
//Memo
"Comments for the ATC analysis"
</SUBDATA>
```

Data ATCExtraMonitor

ATCFlowValue

This subdata section contains a list of a flow values for specified transfer levels. Each line consists of two values: Flow Value (flow on the monitored element) and a Transfer Level (in MW).

Flow Value: Contains a string describing which monitor this belongs to.

Transfer Level: Contains the value for this extra monitor at the last linear iteration.

Example:

```
<SUBDATA ATCFlowValue>  
//MWFlow TransferLevel  
94.05 55.30  
105.18 80.58  
109.02 107.76  
</SUBDATA>
```

Data ATCSenario

TransferLimiter

This subdata section contains a list of the TransferLimiters for this scenario. Each line contains fields relating one of the Transferlimiters. The fields are written out in the following order:

Limiting Element: Contains a description of the limiting element. The possible values are

"PowerFlow Divergence"

"AREA num"

"SUPERAREA name"

"ZONE num"

"BRANCH num1 num2 ckt"

"INJECTIONGROUP name"

"INTERFACE name"

Limiting Contingency: The name of the limiting contingency. If blank, then this means it's a limitation in the base case.

MaxFlow: The transfer limitation in MW.

PTDF: The PTDF on the limiting element in the base case (not in percent).

OTDF: The OTDF on the limiting element under the limiting contingency.

LimitUsed: The limit which was used to determine the MaxFlow in per unit.

PreTransEst: The estimated flow on the line after the contingency but before the transfer in per unit.

MaxFlowAtLastIteration: The total transfer at the last iteration in per unit.

IterativelyFound: Either YES or NO depending on whether it was iteratively determined.

Example:

```
<SUBDATA TransferLimiter >
"BRANCH 40767 42103 1" "Contin" 2.84 -0.0771 -0.3883 -4.35 -4.35 -0.01 "-55.88" YES
"BRANCH 42100 42321 1" "Contin" 4.42 0.1078 0.5466 6.50 5.64 1.57 " 22.59" NO
"BRANCH 42168 42174 1" "Contin" 7.45 -0.0131 -0.0651 -1.39 -1.09 4.60 "-33.31" NO
"BRANCH 42168 42170 1" "Contin" 8.54 0.0131 0.0651 1.39 1.02 5.69 " 26.10" NO
"BRANCH 41004 49963 1" "Contin" 9.17 -0.0500 -0.1940 -4.39 -3.16 6.32 " 68.73" NO
"BRANCH 46403 49963 1" "Contin" 9.53 0.0500 0.1940 4.46 3.16 6.68 "-68.68" NO
"BRANCH 42163 42170 1" "Contin" 10.14 -0.0131 -0.0651 -1.39 -0.92 7.29 "-15.58" NO
</SUBDATA>
```

ATCExtraMonitor

This subdata section contains a list of the ATCExtraMonitors for this scenario. Each line contains three fields relating one of the ATCExtraMonitors. The first field describes the ATCExtraMonitor which this subdata corresponds to. The second and third variables are the initial value and sensitivity

for this extra monitor for the scenario. An option fourth field may be included if we are using one of the iterated ATC solution options. This field must be the String "ATCFlowValue".

Monitor Description: Contains a string describing which monitor this belongs to.

Initial Value: Contains the value for this extra monitor at the last linear iteration.

Sensitivity: Contains the sensitivity of this monitor.

ATCFlowValue: A string which signifies that a block will follow which stores a list of flow values for specified transfer levels. Each line of the block consists of two values: Flow Value (flow on the monitored element) and a Transfer Level (in MW). The block is terminated when a line of text that starts with 'END' is encountered.

Example:

```
<SUBDATA ATCExtraMonitor>
"Interface<KEY1>Left-Right</KEY1>" 40.0735 0.633295
"Branch<KEY1>2</KEY1><KEY2>5</KEY2><KEY3>1</KEY3>" 78.7410 0.266589
</SUBDATA>
```

Data AuxFileExportFormatData

DataBlockDescription

This subdata section is used to define the objects that should be included in an auxiliary file along with their fields, subdata sections, and any filter used to specify which objects should be included. Each line contains the following:

ObjectType: Name of the object to include in the auxiliary file.

[FieldList]: List of fields to include. Must be enclosed in brackets. This list can either be space-delimited or comma-delimited.

[SubdataList]: List of subdata sections to include. This list must be enclosed in brackets and can be either space-delimited or comma-delimited. Include empty brackets to not include subdata or for objects that do not have any subdata sections.

"Filter": Description of the filter to use for determining which objects to include. This must be enclosed in double quotes. If no filter is to be used, empty double quotes should be included. Valid entries are: "", "filtername", "AREAZONE", and "SELECTED".

Example:

```
<SUBDATA DataBlockDescription>
// ObjectType [FieldList] [SubdataList] "Filter"
Area [AreaName, AreaNum] [] "SELECTED"
Gen [BusNum, BusName, GenID] [BidCurve, ReactiveCapability] ""
</SUBDATA>
```

Data AuxFileExportFormatDisplay

DataBlockDescription

This subdata section is used to define the objects that should be included in an auxiliary file along with their fields, subdata sections, and any filter used to specify which objects should be included. Each line contains the following:

ObjectType: Name of the object to include in the auxiliary file.

[FieldList]: List of fields to include. Must be enclosed in brackets. This list can either be space-delimited or comma-delimited.

[SubdataList]: List of subdata sections to include. This list must be enclosed in brackets and can be either space-delimited or comma-delimited. Include empty brackets to not include subdata or for objects that do not have any subdata sections.

"Filter": Description of the filter to use for determining which objects to include. This must be enclosed in double quotes. If no filter is to be used, empty double quotes should be included. Valid entries are: "", "filtername", "AREAZONE", and "SELECTED".

Example:

```
<SUBDATA DataBlockDescription>
// ObjectType [FieldList] [SubdataList] "Filter"
DisplayArea [AreaName, AreaNum, SOAuxiliaryID] [] ""
DisplayTransmissionLine [BusNum, BusNum:1, LineCircuit, SOAuxiliaryID] [Line] "Nominal
Voltage > 138 kV"
</SUBDATA>
```

Data BGCalculatedField

Condition

Bus Group Calculated Fields are used for creating a calculation that operates on the buses in a area, zone, or substation. Part of the definition is a filter which specifies which buses to operate over. This subdata section is identical to the Condition subdata section of the Filter object type.

Data Bus

MWMarginalCostValues**MvarMarginalCostValues****LPOPFMarginalControls**

These three sections contain specific values computed for an OPF solution. In MWMarginalCostValues or MvarMarginalCostValues these specific values are the MW or Mvar marginal prices for each constraint. In LPOPFMarginalControls the values are the sensitivities of the controls with respect to the cost of each bus.

Example:

```
<SUBDATA MWMarginalCostValues>  
//Value  
16.53  
0.00  
21.80  
</SUBDATA>
```

Data BusViewFormOptions

BusViewBusField**BusViewFarBusField****BusViewGenField****BusViewLineField****BusViewLoadField****BusViewShuntField**

The values represent specific fields on the custom defined bus view onelines. Each line contains two values:

Location : The various locations on the customized bus view contain slots for fields. This is the slot number.

FieldDescription : This is a string enclosed in double quotes. The string itself is delimited by the @ character. The string contains five values:

Name of Field: The name of the field. Special fields that appear on dialog by default have special names. Otherwise these are the same as the fieldnames of the AUX file format (for the "other fields" feature on the dialogs).

Total Digit: Number of total digits for a numeric field.

Decimal Points: Number of decimal points for a numeric field.

Color: This is the color of the field. It is not presently used.

Increment Value: This is the "delta per mouse" click for the field.

Example:

```
<SUBDATA BusViewLineField>
0 "MW Flow@6@1@0@0"
1 "MVar Flow@6@1@0@0"
2 "MVA Flow@6@1@0@0"
3 "BusAngle:1@6@2@0@0"
</SUBDATA>
```

Data ColorMap

ColorPoint

A colorpoint is simply described by a real number (between 0 and 100) and an integer describing the color written on a single line of text

cmvalue cmcolor (Example: 100 56648)

cmvalue: Real number between 0 and 100 (minimum to maximum value).

cmcolor: Integer between 0 and 16,777,216. Value is determined by taking the red, green and blue components of the color and assigning them a value between 0 and 255. The color is then equal to $\text{red} + 256 * \text{green} + 256 * 256 * \text{blue}$.

Example:

```
<SUBDATA ColorPoint>
// Value Color
100.0000 127
87.5000 255
62.5000 65535
50.0000 8388479
37.5000 16776960
12.5000 16711680
0.0000 8323072
</SUBDATA>
```

Data Contingency

CTGElementAppend

The format for this data type is the same as described below for CTGElement, with the exception that the elements listed in this data section will be added to the existing list of contingency elements, instead of replace the existing elements.

CTGElement

A contingency element is described by up to four entries. All entries must be on a single line of text.

"Action" "ModelCriteria" Status //comment

Action: String describing the action associated with this element. See below for actions available.

ModelCriteria: This is the name of a ModelFilter or ModelCondition under which this action should be performed. This entry is optional. If it is not specified, then a blank (or no criteria) is assumed. If you want to enter a Status, then use must specify "" as the ModelCriteria.

Status: Three options

CHECK: perform action if ModelCriteria is true.

ALWAYS: perform action regardless of ModelCriteria.

NEVER: do not perform action.

This entry is optional. If it is not specified, then CHECK is assumed.

Comment: All text to the right of the comment symbol (//) will be saved with the CTGElement as a comment.

Possible Actions:

Calling of a name ContingencyBlock

CONTINGENCYBLOCK | name

Calls a ContingencyBlock and executes each of the actions in that block.

Transmission Line or Transformer outage or insertion

BRANCH | bus1# bus2# ckt | OPEN

| CLOSE

Takes branch out of service, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

Series Capacitor Bypass or In Service

SERIESCAP | bus1# bus2# ckt | BYPASS

| INSERVICE

Bypasses a series capacitor, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

DC Transmission Line outage or insertion

DCLine | bus1# bus2# ckt | OPEN
| CLOSE

Takes DC Line out of service, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the dc transmission line.

DC Line set a specific value

DCLINE | bus1# bus2# ckt | SET_P_TO | value | MW
| | CHANGE_P_BY | | PERCENT
| | SET_I_TO | | AMPS
| | CHANGE_I_BY

Use to set the DC line setpoint to a particular value. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the dc transmission line.

Phase Shifter set a specific value

PHASESHIFTER | bus1# bus2# ckt | SET_P_TO | value | MW
| | CHANGE_P_BY | | PERCENT

Use to set the phase shift angle to a particular value. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

3-Winding Transformer outage or insertion

3WXFORMER | bus1# bus2# bus3# ckt | OPEN
| CLOSE

Takes all three windings of a 3-winding transformer out of service, or puts them in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# bus3# ckt] may be replaced by the label of the three winding transformer.

Interface outage or insertion

INTERFACE | name | OPEN
| CLOSE

Takes all monitored branches in the interface out of service, or puts them all in service. Note: the [name] may be replaced by the label of the interface.

Bus outage causes all lines connected to the bus to be outage

BUS | bus# | OPEN

Takes all branches connected to the bus out of service. Also outages all generation, load, or shunts attached to the bus. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

Generator, Load, or Switched Shunt outage or insertion

GEN | bus# id | OPEN
 LOAD | bus# id | CLOSE
 SHUNT | bus#
 INJECTIONGROUP | name

Takes a generator, load, or shunt out of service, or puts it in service. If specifying an injection group, the status of all devices in the injection group will be changed. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the sequence [bus1# ckt] or [name] may be replaced by the label of the device.

Generator, Load or Switched Shunt movement to another bus

GEN | bus1# | MOVE_PQ_TO | bus2# | value | MW
 LOAD | | MOVE_P_TO | | MVAR
 SHUNT | | MOVE_Q_TO | PERCENT

Use to move generation, load or shunt at a bus1 over to bus2. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

Generator, Load or Switched Shunt set or change a specific value

GEN | bus# | SET_P_TO | value | MW
 LOAD | | SET_Q_TO | | MVAR
 SHUNT | | SET_PQ_TO | PERCENT
 | SET_VOLT_TO |
 | CHANGE_P_BY |
 | CHANGE_Q_BY |
 | CHANGE_PQ_BY |
 | CHANGE_VOLT_BY |

Use to set the generation, load, or shunt at a bus to a particular value. Also can use to change by a specified amount. The voltage setpoints only apply to SHUNTS and GENs. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

Injection Group set a specific value

INJECTIONGROUP | name | SET_P_TO | value | MW
 | PERCENT

Use to set the MW generation/load in an injection group to a particular value.

Injection Group change specific value

```
INJECTIONGROUP | name | CHANGE_P_TO | value | MW
| PERCENT
| MWMERITORDER
| PERCENTMERITORDER
```

Use to change the MW generation/load in an injection group by a particular value. Note that MW and PERCENT OPTIONS will change each point in the injection group by a value in proportion to the participations factors of the group. MWMERITORDER and PERCENTMERITORDER will modify points in the injection group by closing or opening points in the group in order of descending participation factors until the total change greater than or equal to the requested change has been achieved. Note: the [name] may be replaced by the label of the injection group.

Execute a Power Flow Solution

```
SOLVEPOWERFLOW
```

Include this action to cause the solution of the contingency to be split into pieces. Actions that are listed before each SOLVEPOWERFLOW call will be performed as a group.

Make-Up Power Compensation.

Only valid immediately following a SET, CHANGE, OPEN or CLOSE action on a Generator, Shunt or Load. This describes how the change in MW or MVAR are picked up by buses throughout the system. The values specify participation factors

```
COMPENSATION
bus#1 value1
bus#2 value2
...
END
```

Example:

```
<SUBDATA CTGElement>
```

```
// just some comments
```

// action	Model Criteria	Status	comment
"BRANCH 40821 40869 1 OPEN"	""	ALWAYS	//Raver - Paul 500 kV
"GEN 45041 1 OPEN"	""	ALWAYS	//Trip Unit #2
"BRANCH 42702 42727 1 OPEN"	"Line X Limited"	CHECK	//Open Fern Hill
"GEN 40221 1 OPEN"	"Interface L1"	CHECK	//Drop ~600 MW
"GEN 40227 1 OPEN"	"Interface L2"	CHECK	//Drop ~1200 MW
"GEN 40221 1 OPEN"	"Interface L3"	CHECK	//Drop ~600 MW
"GEN 40227 1	"Interface L3+"	CHECK	//Drop ~1200

```

OPEN"
</SUBDATA>

```

MW

Note: ContingencyElement object types can also be created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency to which the ContingencyElement belongs.

LimitViol

A LimitViol is used to describe the results of a contingency analysis run. Each Limit Violation lists eight values:

ViolType ViolElement Limit ViolValue PTFD PTDF InitialValue Reason

ViolType: One of five values describing the type of violation

BAMP: branch amp limit violation.

BMVA: branch MVA limit violation.

VLOW: bus low voltage limit violation.

VHIGH: bus high voltage limit violation.

INTER: interface MW limit violation.

ViolElement: This field depends on the ViolType.

for VLOW, VHIGH: "bus1#" or "busname_buskv" or "buslabel"

for INTER: "interfacename" or "interfacelabel"

for BAMP, BMVA: "bus1# bus2# ckt violationbus# MWFlowDirection"

violationbus# is the bus number for the end of the branch which is violated

MWFlowDirection is the direction of the MW flow on the line. Potential values are "FROMTO" or "TOFROM"

Note: each bus# may be replaced with the name underscore nominal kV string enclosed in single quotes, or bus# values may be replaced by a string enclosed in single quotes representing the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

Limit: This is the numerical limit which was violated.

ViolValue: This is the numerical value of the violation.

PTDF: This field is optional. It only makes sense for interface or branch violations. It stores a sensitivity of the flow on the violating element during in the base case with respect to a transfer direction This must be calculated using the Contingency Analysis Other Actions related to Sensitivities.

OTDF: This field is optional. It only makes sense for interface or branch violations. It stores a sensitivity of the flow on the violating element during this contingency with respect to a transfer direction. This must be calculated using the Contingency Analysis Other Actions related to Sensitivities.

InitialValue: This stores a number. This stores the base case value for the element which is being violated. This is used to compare against when looking at change violations.

Reason: This will say whether this was a pure violation or is being reported as a violation because the change from the base case is higher than a specified threshold.

LIMIT - means this is a violation of a line/interface/bus limit

CHANGE - means this is being reported as a limit because the change from the initial value is higher than allowed

Example:

```
<SUBDATA LimitViol>
```

```
BAMP "1 3 1 1 FROMTO" 271.94031 398.48096 10.0 15.01 // Note OTDF/PTDF  
INTER "Right-Top" 45.00000 85.84451 // Note OTDF/PTDF not included  
</SUBDATA>
```

Note: ViolationCTG object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency to which the ViolationCTG belongs.

Sim_Solution_Options

These describe the power flow solution options which should be used under this particular contingency. The format of the subdata section is two lines of text. The first line is a list of the fieldtypes for Sim_Solution_Options which should be changed. The second line is a list of the values. Note that in general, power flow solution options are stored at three different locations in contingency analysis. When implementing a contingency, Simulator gives precedence to these three locations in the following order:

1. Contingency Record Options (stored with the particular contingency)
2. Contingency Tool Options (stored with CTG_Options)
3. The global solution options

WhatOccurredDuringContingency

Each line of this subdata section is part of a text description of what actually ended up being implemented for this contingency. This will list which actions were executed and which actions ended up being skipped because of their model criteria. Each line of the subdata section must be enclosed in quotes.

Data CTG Options

Sim_Solution_Options

These describe the power flow solution options which should be used under this particular contingency. The format of the subdata section is two lines of text. The first line is a list of the fieldtypes for Sim_Solution_Options which should be changed. The second line is a list of the values. Note that in general, power flow solution options are stored at three different locations in contingency analysis. When implementing a contingency, Simulator gives precedence to these three locations in the following order:

- Contingency Record Options (stored with the particular contingency)
- Contingency Tool Options (stored with CTG_Options)
- The global solution options

Data CTGElementBlock

CTGElement

This format is the same as for the Contingency objecttype, however, you can not call a ContingencyBlock from within a contingencyblock.

CTGElementAppend

When a subdata section is defined as **CTGElementAppend** rather than **CTGElement**, the actions of this subdata section will be appended to the contingency actions, instead of replacing them. This format is the same as for the Contingency objecttype, however, you can not call a ContingencyBlock from within a contingencyblock.

Note: CTGElementBlockElement object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency block to which the CTGElementBlockElement belongs.

Data CustomCaseInfo

ColumnInfo

Each line of this SUBDATA section can be used for specifying the column width of particular columns of the respective Custom Case Information Sheet. The line contains two values – the column and then a column width. This is shown in the following example.

Example:

```
<SUBDATA ColumnInfo>  
"SheetCol" 133  
"SheetCol:1" 150  
"SheetCol:2" 50  
</SUBDATA>
```

Data CustomColors

CustomColors

These describe the customized colors used in Simulator, which are specified by the user. A custom color is an integer describing a color. Each custom color is written on a single line of text and is an integer between 0 and 16,777,216. The value is determined by taking the red, green, and blue components of the color and assigning them a value between 0 and 255. The color is then equal to $\text{red} + 256 * \text{green} + 256 * 256 * \text{blue}$. Each line contains only one integer that corresponds to the color specified.

Example:

```
<SUBDATA CustomColors>  
9823301  
8613240  
</SUBDATA>
```

Data DataGrid

ColumnInfo

Contains a description of the columns which are shown in the respective data grid. Each line of text contains four fields: **VariableName, ColumnWidth, TotalDigits, DecimalPoints**

VariableName: Contains the variable which is shown in this column.

ColumnWidth: The column width.

TotalDigits: The total digits displayed for numerical values.

DecimalPoints: The decimal points shown for numerical values.

Example:

```
DATA (DataGrid, [DataGridName])
```

```
{
```

```
  BUS
```

```
  <SUBDATA COLUMNINFO>
```

```
  BusNomVolt 100 8 2
```

```
  AreaNum 50 8 2
```

```
  ZoneNum 50 8 2
```

```
  </SUBDATA>
```

```
  BRANCHRUN
```

```
  <subdata COLUMNINFO >
```

```
  BusNomVolt: 0 100 8 2
```

```
  BusNomVolt: 1 100 8 2
```

```
  LineMW: 0 100 9 3
```

```
  </SUBDATA>
```

```
}
```

Data DynamicFormatting

DynamicFormattingContextObject

This subdata section contains a list of the display object types which are chosen to be selected. Each line of the section consists of the following:

DisplayObjectType: The object type of the display object. These are generally the same as the values seen in the subdata section `SelectByCriteriaSetType` of `SelectByCriteriaSet` object types. The only exception is the string *CaseInfo*, which is used for formatting applying to the case information displays.

(WhichFields) : For display objects that can reference different fields, this sets which of those fields it should select (e.g. select only Bus Name Fields). The value may be either ALL or SPECIFIED.

(ListOfFields) : If WhichFields is set to SPECIFIED, then a delimited list of fields follows.

Example:

```
<SUBDATA DynamicFormattingContextObject>
// Note: CaseInfo applies to case information displays
CaseInfo "SPECIFIED" BusName
DisplayAreaField "ALL"
DisplayBus
DisplayBusField "SPECIFIED" BusName BusPUVolt BusNum
DisplayCircuitBreaker
DisplaySubstation
DisplaySubstationField "SPECIFIED" SubName SubNum BusNomVolt BGLoadMVR
DisplayTransmissionLine
DisplayTransmissionLineField "ALL"
</SUBDATA>
```

LineThicknessLookupMap

LineColorLookupMap

FillColorLookupMap

FontColorLookupMap

FontSizeLookupMap

BlinkColorLookupMap

XoutColorLookupMap

FlowColorLookupMap

SecondaryFlowColorLookupMap

The values of the lookup table for the characteristics that can be modified by the dynamic formatting tool. The first line contains the two following fields:

fieldname: It is the field that the lookup table is going to look for.

usediscrete: Set to YES or NO. If set to YES, the characteristic values will be discrete, meaning that the characteristic value will correspond exactly to the one specified in the table. If set to NO, the characteristic values will be continuous, which means the characteristic value will be an interpolation of the high and low closest values specified in the table.

The following lines contain two fields:

fieldvalue: The value for the field.

characteristicvalue: The corresponding characteristic value for such field value.

Example:

```
<SUBDATA LineColorLookupMap>
// FieldName UseDiscrete
BusPUVolt YES
// FieldValue Color
1.02 16711808
1.05 8454143
1.1 16744703
</SUBDATA>
```

Data Filter

Condition

Conditions store the conditions of the filter. Each condition is described by one line of text which can contain up to five fields.

Variablename condition "value" ("othervalue") (fieldopt)

Variablename: It is one of the fields for the object_type specified. It may optional be followed by a colon and a non-negative integer. If not specified, 0 is assumed.

Example: on a LINE, 0 = from bus, 1 = to bus

Thus: LineMW: 0 = the MW flow leaving the from bus

LineMW: 1 = the MW flow leaving the to bus

Condition: Possible Values Alternate1 Alternate2 Requirements

between >< requires other value

notbetween ~>< requires other value

equal = ==

notequal <> ~=

greaterthan >

lessthan <

greaterthanorequal >=

lessthanorequal <=

contains

notcontains

startswith

notstartswith

inrange

notinrange

meets

notmeets

isblank

notisblank

value: The value to compare to.

For fields associated with strings, this must be a string

For fields associated with real numbers, this must be a number.

For fields associated with integers, this is normally an integer, except when the Condition is "inrange" or "notinrange". In this case, value is a comma/dash separated number string.

(othervalue): If required, the other value to compare to.

(FieldOpt): Optional integer value with following meanings.

0 - strings are case insensitive, use number fields directly

(0 is the default value if not otherwise specified)
 1 - strings are case sensitive, take ABS of field values

Example:

```
DATA (FILTER, [objecttype, filtername, filterlogic, filterpre])
{
BUS "a bus filter" "AND" "no"
<SUBDATA CONDITION>
BusNomVolt > 100
AreaNum inrange "1-5 , 7 , 90-95"
ZoneNum between
</SUBDATA>

BRANCH "a branch filter" "OR" "no"
<subdata CONDITION>
BusNomVolt: 0 > 100 // Note location 0 means from bus
BusNomVolt: 1 > 100 // Note location 1 means to bus
LineMW: 0 > 100 1 // Note, final field 1 denotes absolute value
</SUBDATA>
}
```

Data Gen

BidCurve

BidCurve subdata is used to define a piece-wise linear cost curve (or a bid curve). Each bid point consists of two real numbers on a single line of text: a MW output and then the respective bid (or marginal cost).

Example:

```
<SUBDATA BidCurve>
// MW Price[$/MWhr]
100.00 10.6
200.00 12.4
400.00 15.7
500.00 16.0
</SUBDATA>
```

ReactiveCapability

Reactive Capability subdata is used to the reactive capability curve of the generator. Each line of text consists of three real numbers: a MW output, and then the respective Minimum MVAR and Maximum MVAR output.

Example:

```
<SUBDATA ReactiveCapability>
// MW MinMVAR MaxMVAR
100.00 -60.00 60.00
200.00 -50.00 50.00
400.00 -30.00 20.00
500.00 -5.00 2.00
</SUBDATA>
```

Note: ReactiveCapability object types can also be directly created inside their own DATA section as well. Two of the key fields of the object are then the bus number and generator ID of the generator to which the ReactiveCapability point belongs.

Data GeoDataViewStyle

TotalAreaValueMap

This subdata section is used to define the lookup table for determining the total area size of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Total Area attribute.

TotalArea: The total area size of the object.

Example:

```
<SUBDATA TotalAreaValueMap>
// FieldValue TotalArea
1.000 0
4.000 23
7.000 46
</SUBDATA>
```

RotationRateValueMap

This subdata section is used to define the lookup table for determining the rotation rate of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Rotation Rate attribute.

RotationRate: The rotation rate of the object. Entered in Hz.

Example:

```
<SUBDATA RotationRateValueMap>
// FieldValue RotationRate
1.000 0.00
4.000 0.10
7.000 0.20
</SUBDATA>
```

RotationAngleValueMap

This subdata section is used to define the lookup table for determining the rotation angle of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Rotation Angle attribute.

RotationAngle: The rotation angle of the object. Entered in degrees.

Example:

```
<SUBDATA RotationAngleValueMap>
```

```
// FieldValue RotationAngle  
1.000 -90.0  
4.000 0.0  
7.000 90.0  
</SUBDATA>
```

LineThicknessValueMap

This subdata section is used to define the lookup table for determining the thickness of the border line around geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Line Thickness attribute.

LineThickness: The line thickness of the border line around the object. This should be an integer value.

Example:

```
<SUBDATA LineThicknessValueMap>  
// FieldValue LineThickness  
1.000 1  
4.000 2  
7.000 3  
</SUBDATA>
```

Data GlobalContingencyActions

CTGElementAppend

This format is the same as for the Contingency objecttype

CTGElement

This format is the same as for the Contingency objecttype

Note: GlobalContingencyActionsElement object types can also be directly created inside their own DATA section as well.

Data HintDefValues

HintObject

Stores the values for the custom hints. Each line has one value:

FieldDescription: This is a string enclosed in double quotes. The string itself is delimited by the @ character. The string contains five values:

Name of Field: The name of the field. Special fields that appear on dialog by default have special names. Otherwise these are the same as the fieldnames of the AUX file format (for the "other fields" feature on the dialogs).

Total Digit: Number of total digits for a numeric field.

Decimal Points: Number of decimal points for a numeric field.

Include Suffix: Set to 0 for not including the suffix, and set to 1 to include it.

Field Prefix: The prefix text.

Example:

```
<SUBDATA HintObject>  
"BusPUVolt@4@1@1@PU Volt ="  
"BusAngle@4@1@1@Angle ="  
</SUBDATA>
```

Data InjectionGroup

PartPoint

A participation point is used to describe the contents of an injection group. Each participation point lists six values:

PointType PointBusNum PointID PointParFac ParFacCalcType ParFacNotDynamic

PointType: One of two values describing the type of violation:

GEN: a generator

LOAD: a load

INJECTIONGROUP: another injection group

PointBusNum: The bus number of the partpoint if the type is GEN or LOAD. Value will be blank for an injection group type. Note: bus# values may be replaced by a string enclosed in double quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in double quotes that represents the label of the bus or a string representing the label of the generator or load.

PointID: For a GEN or LOAD type, this is the id for the partpoint. For an INJECTIONGROUP type, this is the name or label of the injection group.

PointParFac: The participation factor for the point.

ParFacCalcType: How the participation point is calculated. There are several options depending on the PointType.

Generators: SPECIFIED, MAX GEN INC, MAX GEN DEC, or MAX GEN MW

Loads: SPECIFIED or LOAD MW

Injection Groups: SPECIFIED

ParFacNotDynamic: Should the participation factor be recalculated dynamically as the system changes.

Example:

```
<SUBDATA PartPoint
"GEN" 1 "1" 1.00 "SPECIFIED" "NO"
"GEN" 2 "1" 5.00 "SPECIFIED" "NO"
"GEN" 4 "1" 104.96 "MAX GEN INC" "NO"
"GEN" 6 "1" 50.32 "MAX GEN DEC" "YES"
"GEN" 7 "1" 600.00 "MAX GEN MW" "NO"
"LOAD" 2 "1" 5.00 "SPECIFIED" "NO"
"LOAD" 5 "1" 130.00 "LOAD MW" "NO"
"LOAD" 6 "1" 200.00 "LOAD MW" "YES"
</SUBDATA>
```

Note: PartPoint object types can also be directly created inside their own DATA section as well. One of the key fields of the PartPoint object is then the name of the injection group to which the participation point belongs.

Data Interface

InterfaceElement

An interface's subdata contains a list of the elements in the interface. Each line contains a text description of the interface element. Note that this text description must be encompassed by quotation marks. There are five kinds of elements allowed in an interface. Please note that the direction specified in the monitoring elements is important.

"BRANCH num1 num2 ckt" Monitor the MW flow on the branch starting from bus num1 going to bus num2 on the branch this branch. (order of bus numbers defines the direction)

"AREA num1 num2" Monitor the sum of the AC tie line MW flows from area num1 to area num2.

"ZONE num1 num2" Monitor the sum of the AC tie line MW flows from zone num1 to zone num2.

"BRANCHOPEN num1 num2 ckt" When monitoring the elements in this interface, monitor them under the contingency of opening this branch.

"BRANCHCLOSE num1 num2 ckt" When monitoring the elements in this interface, monitor them under the contingency of closing this branch.

"DCLINE num1 num2 ckt" Monitor the flow on a DC line.

"INJECTIONGROUP 'name'" Monitor the net injection from an injection group (generation contributes a positive injection, loads negative).

"GEN num1 id" Monitor the net injection from a generator (output is positive injection).

"LOAD num1 id" Monitor the net injection from a load (output is negative injection).

"MSLINE num1 num2 ckt" Monitor the MW flow on the multi-section line starting from bus num1 going to bus num2 with circuit ckt.

"INTERFACE 'name'" Monitor the MW flow on the interface given by name.

Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. Labels may also be used as follows:

- bus# values for all element types may be replaced by a string enclosed in single quotes where the string is the label of the bus
- for GEN or LOAD elements, the section *num1 id* may be replaced by the device's label
- for MSLINE, DCLINE, or BRANCH elements, the *num1 num2 ckt* section may be replaced by the device's label

For the interface element type "BRANCH num1 num2 ckt" and "DCLINE num1 num2 ckt", an optional field can also be written specifying whether the flow should be measured at the far end. This field is either YES or NO.

Example:

```
<SUBDATA InterfaceElement
```

```
"BRANCH 8 9 1" NO // monitor the flow from bus 8 to bus 9 on this branch
```

```
"BRANCH 12 33 1" YES // monitor the flow from bus 12 to bus 33 on branch
```

```
// measurefarend is set to true, therefore, we are
// monitoring the MW flow that arrives at bus 33

"AREA 2 1" // monitor tie line flow from area 2 to area 1
"ZONE 66 53" // monitor tie lines flows from zone 66 to zone 53
"BRANCHOPEN 5 6 1" // doe monitoring after branch opens
"BRANCHCLOSE 7 10 1" // doe monitoring after branch closes
</SUBDATA>
```

Note: InterfaceElement object types can also be directly created inside their own DATA section as well. One of the key fields of the InterfaceElement object is then the name of the interface to which the interface element belongs.

Data LimitSet

LimitCost

LimitCost records describe the piece-wise unenforceable constraint cost records for use by unenforceable line/interface limits in the OPF or SCOPF. Each row contains two values

PercentLimit: Percent of the transmission line limit.

Cost: Cost used at this line loading percentage value.

Example:

```
<SUBDATA LimitCost>  
//Percent Cost [$/MWhr]  
100.00 50.00  
105.00 100.00  
110.00 500.00  
</SUBDATA>
```

Data Load

BidCurve

BidCurve subdata is used to define a piece-wise linear benefit curve (or a bid curve). Each bid point consists of two real numbers on a single line of text: a MW output and then the respective bid (or marginal cost). These costs must be increasing for loads.

Example:

```
<SUBDATA BidCurve>  
// MW Price[$/MWhr]  
100.00 16.0  
200.00 15.7  
400.00 12.4  
500.00 10.6  
</SUBDATA>
```

Data LPVariable

LPVariableCostSegment

Stores the cost segments for the LP variables. Each line contains four values:

Cost (Up): Cost associated with increasing the LP variable.

Minimum value: Minimum limit of the LP variable.

Maximum value: Maximum limit of the LP variable.

Artificial: Whether the cost segment is artificial or not.

Data ModelCondition

Condition

ModelConditions are the combination of an object and a Filter. They are used to return when the particular object meets the filter specified. As a results, the subdata section here is identical to the Condition subdata section of a [Filter](#). See the description there.

Data ModelExpression

LookupTable

LookupTables are used inside Model Expressions sometimes. These lookup table represent either one or two dimensional tables. If the first string in the SUBDATA section is "x1x2", the this signals that it is a two dimensional lookup table. From that point on it will read the first row as "x2" lookup points, and the first column in the remainder of the rows as the x1 lookup values.

Example:

```
DATA (MODELEXPRESSION, [CustomExpression,ObjectType,CustomExpressionStyle,
CustomExpressionString,WhoAmI,VariableName,WhoAmI:1,VariableName:1], AUXDEF)
{
// The following demonstrated a one dimensional lookup table
22.0000, "oneD", "Lookup", "",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2>",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>",
"", ""
<SUBDATA LookupTable>
// because it does not start with the string x1x2 this will
// represent a one dimensional lookup table
x1 value
0.000000 1.000000
11.000000 22.000000
111.000000 222.000000
</SUBDATA>

0.0000, "twod", "Lookup", "",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2>",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>",
"Gen<KEY1>6</KEY1><KEY2>1</KEY2>",
"Gen<KEY1>6</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>"
<SUBDATA LookupTable>
// because this starts with x1x2 this represent a two dimensional
// lookup table. The first column represents lookup values for x1.
// The first row represents lookup values for x2
x1x2 0.100000 0.300000 // these are lookup heading for x2
0.000000 1.000000 3.000000
11.000000 22.000000 33.000000
111.000000 222.000000 333.000000
</SUBDATA>
}
```

Data ModelFilter

ModelCondition

A Model Filter's subdata contains a list of each ModelCondition in the it. Because a list of Model Conditions is stored within Simulator, this subdata section only stores the name of each ModelCondition on each line.

Example:

```
<SUBDATA ModelCondition>  
  "Name of First Model Condition"  
  "Name of Second Model Condition"  
  "Name of Third Model Condition"  
</SUBDATA>
```

Data MTDCRecord

MTDCBus

For this SUBDTA section, each DC Bus is described on a single line of text with exactly 8 fields specified.

DCBusNum: The number of the DC Bus. Note the DC bus numbers are independent of the AC bus numbers.

DCBusName: The name of the DC bus enclosed in quotes.

ACTerminalBus: The AC terminal to which this DC bus is connected (via a MTDCConverter). If the DC bus is not connected to any AC buses, then specify as zero. You may also specify this as a string enclosed in double quotes with the bus name followed by an underscore character, following by the nominal voltage of the bus.

DCResistanceToground: The resistance of the DC bus to ground. Not used by Simulator.

DCBusVoltage: The DC bus voltage in kV.

DCArea: The area that this DC bus belongs to.

DCZone: The zone that this DC bus belongs to.

DCowner: The owner that this DC bus belongs to.

Note: MTDCBus object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCBus belongs.

MTDCConverter

For this SUBDTA section, each AC/DC Converter is described by exactly 24 field which may be spread across several lines of text. Simulator will keep reading lines of text until it finds 24 fields. All text to the right of the 24th field (on the same line of text) will be ignored. The 24 fields are listed in the following order:

BusNum: AC terminal bus number.

MTDCNBridges: Number of bridges for the converter.

MTDCConvEBas: Converter AC base voltage.

MTDCConvAngMxMn: Converter firing angle.

MTDCConvAngMxMn:1: Converter firing angle max.

MTDCConvAngMxMn:2: Converter firing angle min.

MTDCConvComm: Converter commutating resistance.

MTDCConvComm:1: Converter commutating reactance.

MTDCConvXFRat: Converter transformer ratio.

MTDCFixedACTap: Fixed AC tap.

MTDCConvTapVals: Converter tap.

MTDCConvTapVals:1: Converter tap max.

MTDCConvTapVals:2: Converter tap min.

MTDCConvTapVals:3: Converter tap step size.

MTDCConvSetVL: Converter setpoint value (current or power).

MTDCConvDCPF: Converter DC participation factor.

MTDCConvMarg: Converter margin (power or current).

MTDCConvType: Converter type.

MTDCMaxConvCurrent: Converter Current Rating.

MTDCConvStatus: Converter Status.

MTDCConvSchedVolt: Converter scheduled DC voltage.

MTDCConvIDC: Converter DC current.

MTDCConvPQ: Converter real power.

MTDCConvPQ:1: Converter reactive power.

Note: MTDCConverter object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCConverter belongs.

MTDCTransmissionLine

For this SUBDATA section, each DC Transmission Line is described on a single line of text with exactly 5 fields specified:

DCFromBusNum: From DC Bus Number.

DCToBusNum: To DC Bus Number.

CKTID: The DC Circuit ID.

Resistance: Resistance of the DC Line in Ohms.

Inductance: Inductance of the DC Line in mHenries (Not used by Simulator).

Example:

```
DATA (RECORD, [Num,Mode,ControlBus])
{
//-----
// The first Multi-Terminal DC Transmission Line Record
//-----
1 "Current" "SYLMAR3 (26098)"
  <SUBDATA Bus>
//-----
// DC Bus data must appear on a single line of text
// The data consists of exactly 8 values
// DC Bus Num, DC Bus Name, AC Terminal Bus, DC Resistance to ground,
// DC Bus Voltage, DC Bus Area, DC Bus Zone, DC Bus Owner
//-----
3 "CELILO3P" 0 9999.00 497.92 40 404 1
4 "SYLMAR3P" 0 9999.00 439.02 26 404 1
7 "DC7" 41311 9999.00 497.93 40 404 1
8 "DC8" 41313 9999.00 497.94 40 404 1
9 "DC9" 26097 9999.00 439.01 26 404 1
10 "DC10" 26098 9999.00 439.00 26 404 1
  </SUBDATA>
  <SUBDATA Converter>
```

```
//-----
// convert subdata keeps reading lines of text until it has found
// values specified for 24 fields. This can span any number of lines
// any values to the right of the 24th field found will be ignored
// The next converter will continue on the next line.
//-----
41311 2 525.00 20.25 24.00 5.00 0.0000 16.3100
0.391048 1.050000 1.000000 1.225000 0.950000 0.012500
1100.0000 1650.0000 0.0000 "Rect" 1650.0000 "Closed"
497.931 1100.0000 547.7241 295.3274
41313 4 232.50 15.36 17.50 5.00 0.0000 7.5130
0.457634 1.008700 1.030000 1.150000 0.990000 0.010000
2000.0000 2160.0000 0.1550 "Rect" 2160.0000 "Closed"
497.940 2000.0000 995.8800 561.8186
26097 2 230.00 20.90 24.00 5.00 0.0000 16.3100
0.892609 1.000000 1.100000 1.225000 0.950000 0.012500
-1100.0000 1650.0000 "" "Inv" 1650.0000 "Closed"
439.009 1100.0000 -482.9099 274.5227
26098 4 232.00 17.51 20.00 5.00 0.0000 7.5130
0.458621 1.008700 1.100000 1.120000 0.960000 0.010000
439.0000 2160.0000 "" "Inv" 2160.0000 "Closed"
439.000 1999.9999 -878.0000 544.2775
</SUBDATA>
<SUBDATA TransmissionLine>
//-----
// DC Transmission Segment information appears on a single line of
// text. It consists of exactly 5 value
// From DCBus, To DCBus, Circuit ID, Line Resistance, Line Inductance
//-----
3 4 "1" 19.0000 1300.0000
7 3 "1" 0.0100 0.0000
8 3 "1" 0.0100 0.0000
9 4 "1" 0.0100 0.0000
10 4 "1" 0.0100 0.0000
</SUBDATA>

//-----
// A second Multi-Terminal DC Transmission Line Record
//-----
2 "Current" "SYLMAR4 (26100)"
```

```

<SUBDATA Bus>
5 "CELILO4P" 0 9999.00 497.92 40 404 1
6 "SYLMAR4P" 0 9999.00 439.02 26 404 1
11 "DC11" 41312 9999.00 497.93 40 404 1
12 "DC12" 41314 9999.00 497.94 40 404 1
13 "DC13" 26099 9999.00 439.01 26 404 1
14 "DC14" 26100 9999.00 439.00 26 404 1
</SUBDATA>
<SUBDATA Converter>
41312 2 525.00 20.26 24.00 5.00 0.0000 16.3100
0.391048 1.050000 1.000000 1.225000 0.950000 0.012500
1100.0000 1650.0000 0.0000 "Rect" 1650.0000 "Closed"
497.931 1100.0000 547.7241 295.3969
41314 4 232.50 15.45 17.50 5.00 0.0000 7.5130
0.457634 1.008700 1.030000 1.150000 0.990000 0.010000
2000.0000 2160.0000 0.1550 "Rect" 2160.0000 "Closed"
497.940 2000.0000 995.8800 562.9448
26099 2 230.00 20.90 24.00 5.00 0.0000 16.3100
0.892609 1.000000 1.100000 1.225000 0.950000 0.012500
-1100.0000 1650.0000 "" "Inv" 1650.0000 "Closed"
439.009 1100.0000 -482.9099 274.5227
26100 4 232.00 17.51 20.00 5.00 0.0000 7.5130
0.458621 1.008700 1.100000 1.120000 0.960000 0.010000
439.0000 2160.0000 "" "Inv" 2160.0000 "Closed"
439.000 1999.9999 -878.0000 544.2775
</SUBDATA>
<SUBDATA TransmissionLine>
5 6 "1" 19.0000 1300.0000
11 5 "1" 0.0100 0.0000
12 5 "1" 0.0100 0.0000
13 6 "1" 0.0100 0.0000
14 6 "1" 0.0100 0.0000
</SUBDATA>
}

```

Note: MTDCTransmissionLine object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCTransmissionLine belongs.

Data MultiSectionLine

Bus

A multi section line's subdata contains a list of each dummy bus, starting with the one connected to the From Bus of the MultiSectionLine and proceeding in order to the bus connected to the To Bus of the Line. Note: bus# values may be replaced by a string enclosed in double quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus.

Example:

```
//-----
// The following describes a multi-section line that connects bus
// 2 - 1 - 5 - 6 - 3
//-----
DATA (MultiSectionLine, [BusNum, BusName, BusNum: 1, BusName: 1,
  LineCircuit, MSLineNSections, MSLineStatus] )
{
2 "Two" 3 "Three" "&1" 2 "Closed"
  <SUBDATA Bus>
  1
  5
  6
  </SUBDATA>
}
```

Data Nomogram

InterfaceElementA**InterfaceElementB**

InterfaceElementA values represent the interface elements for the first interface of the nomogram. InterfaceElementB values represent the interface elements for the second interface of the nomogram. The format of these SUBDATA sections is identical to the format of the InterfaceElement SUBDATA section of a normal Interface.

NomogramBreakPoint

This subdata section contains a list of the vertex points on the nomogram limit curve.

Example:

```
<SUBDATA NomogramBreakPoint>  
// LimA LimB  
-100 -20  
-100 100  
80 50  
60 -10  
</SUBDATA>
```

Data NomogramInterface

InterfaceElement

This follows the same convention as the InterfaceElement SUBDATA section described with the [Interface](#) objecttype.

Data Owner

Bus

This subdata section contains a list of the buses which are owned by this owner. Each line of text contains the bus number. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus.

Example:

```
<SUBDATA Bus>
1
35
65
</SUBDATA>
```

Load

This subdata section contains a list of the loads which are owned by this owner. Each line of text contains the bus number followed by the load id. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the bus and load id, the label of the load enclosed in double quotes may be used.

Example:

```
<SUBDATA Load>
5 1 // shows ownership of the load at bus 5 with id of 1
423 1
</SUBDATA>
```

Gen

This subdata section contains a list of the generators which are owned by this owner and the fraction of ownership. Each line of text contains the bus number, followed by the gen id, followed by an integer showing the fraction of ownership. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the bus and generator id, the label of the generator enclosed in double quotes may be used.

Example:

```
<SUBDATA Gen>
78 1 50 // shows 50% ownership of generator at bus 78 with id of 1
23 3 70
```

```
</SUBDATA>
```

Branch

This subdata section contains a list of the branches which are owned by this owner and the fraction of ownership. Each line of text contains the from bus number, followed by the to bus number, followed by the circuit id, followed by an integer showing the fraction of ownership. As an alternative to specifying the bus numbers, strings enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the two numbers and a circuit id, the label of the branch enclosed in double quotes may be used.

Example:

```
<SUBDATA Branch>
```

```
6 10 1 50 // shows 50% ownership of line from bus 6 to 10, circuit 1
```

```
</SUBDATA>
```

Data PieChartGaugeStyle

ColorMap

This is a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values:

Percentage: This is the percentage at which the following scalar and color should be applied.

Scalar: A scalar (multiplier) on the size of the pie chart/gauge.

Color: A color for the pie chart/gauge.

Example:

```
<SUBDATA ColorMap>  
//Percentage Scalar Color  
85.0000 1.5000 33023  
100.0000 2.0000 255  
</SUBDATA>
```

Data PostPowerFlowActions

CTGElementAppend

This format is the same as for the Contingency objecttype.

CTGElement

This format is the same as for the Contingency objecttype.

Note: PostPowerFlowActionsElement object types can also be directly created inside their own DATA sections as well.

Data PWCaseInformation

PWCaseHeader

This subdata section contains the Case Description in free-formatted text. Note: as it is read back into Simulator all spaces from the start of each line are removed.

Data PWFormOptions

PieSizeColorOptions

There can actually be several PieSizeColorOptions subdata sections for each PWFormOptions object. The first line of each subdata section, the first line of text consist of exactly four values

ObjectName: The objectname of the type of object these settings apply to. Will be either be BRANCH or INTERFACE.

FieldName: The fieldname for the pie charts that these settings apply to.

UseDiscrete: Set to YES to use a discrete mapping of colors and size scalars instead of interpolating for intermediate values.

UseOtherSettings: Set to YES to default these settings to the BRANCH MVA values for BRANCH object. This allows you to apply the same settings to all pie charts.

After this first line of text, if the UseOtherSettings Value is NO, then another line of text will contain exactly three values:

ShowValue: This is the percentage at which the value should be drawn on the pie chart.

NormalSize: This is the scalar size multiplier which should be used for pie charts below the lowest percentage specified in the lookup table.

NormalColor: This is the color which should be used for pie charts below the lowest percentage specified in the lookup table.

Finally the remainder of the subdata section will contain a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values

Percentage: This is the percentage at which the follow scalar and color should be applied.

Scalar: A scalar (multiplier) on the size of the pie charts.

Color: A color for the pie charts.

Data PWLPOPFGViol

OPFControlSense**OPFBusSenseP****OPFBusSenseQ**

This stores the control sensitivities for each contingency violation during OPF/SCOPF analysis. Each line contains one value:

Sensitivity: The value of the sensitivity with respect to each control in OPFControlSense or with respect to each bus in OPFBusSenseP and OPFBusSenseQ.

Example:

```
<SUBDATA OPFControlSense>  
// Value  
1.000441679  
2.447185E-7  
-1.1109307E-6  
1.6427327E-7  
0  
</SUBDATA>
```

Data PWLPTabRow

LPBasisMatrix

This subdata section stores the basis matrix associated with the final LP OPF solution. Each line contains two values:

Variable: The basic variable.

Value: The sensitivity of the constraint to the basic variable.

Example:

```
<SUBDATA LPBasisMatrix>
// Var Value
1 1.00000
2 1.00000
5 1.00000
6 1.00000
</SUBDATA>
```

Data PWPVResultListContainer

PWPVResultObject

This subdata section contains the results of a particular PV Curve scenario (a particular contingency or the base case). The data consists of two general sections: the first three rows of text contain the "independent axis" of the PV Curve. The first row starts with the string INDNOM and is followed by a list of numbers representing the nominal shift, the second row starts with INDEXP and is followed by the export shift, and the third row starts with INDIMP and is followed by the import shift. Following these rows is a list of all the tracked quantities. Each tracked quantity row consists of three parts which are separated by the strings `?f=` and `&v=`. The first part of the string represents a description of the power system object being tracked, the second part represents the field name being tracked, and the third contains a list of all the values at the various shift levels.

Example:

```
<SUBDATA PWPVResultObject>
INDNOM 0.00 500.00 1000.00 1500.00 1750.00 1875.00 1975.00
INDEXP 0.00 500.00 1000.00 1500.00 1750.00 1875.00 1975.00
INDIMP 0.00 -417.23 -701.58 -890.58 -952.60 -975.35 -990.43
Bus '3'?f=BusPUVolt&v= 0.993 0.983 0.964 0.939 0.926 0.919 0.914
Bus '5'?f=BusPUVolt&v= 1.007 1.000 0.982 0.956 0.940 0.932 0.926
Gen '4' '1'?f=GenMVR&v= 19.99 245.27 523.62 831.13 986.84 1060.6 1118.7
Gen '6' '1'?f=GenMVR&v= -6.59 -120.84 -131.37 -39.53 48.35 103.8 154.5
</SUBDATA>
```

Data QVCurve

QVPoints

This subdata section contains a list of the QV Curve points calculated for the respect QVCurve. Each line consists of exactly four values:

PerUnitVoltage: The per unit voltage of the bus for a QV point.

FictitiousMvar: The amount of Mvar injection from the fictitious generator at this QV point.

ShuntDeviceMvar: The Mvar injection from any switched shunts at the bus.

TotalMvar: The total Mvar injection from switched shunts and the fictitious generator.

ReservesMvar: Total amount of Mvar reserves available at the bus.

ReservesTotalMvar: Total Mvar injection from the switched shunts, fictitious generator, and available reserves.

Example:

```
DATA (QVCURVE, [BusNum,CaseName,qv_VQ0,qv_Q0,qv_Vmax,qv_QVmax,qv_VQmin,qv_Qmin,
qv_Vmin,qv_QVmin,Qinj_Vmax,Qinj_0,Qinj_min,Qinj_Vmin])
{
5 "BASECASE" 0.880 0.000 1.100 312.490 0.480 -221.072
0.180 -86.334 191.490 -77.373 -244.075 -89.562
<SUBDATA QVPoints>
// NOTE: This bus has a constant impedance
// switched shunt value of -100 Mvar at it.
// V(PU), Q(MVR), Q_shunt(MVR), Q_tot(MVR), Q_res(MVR), Q_tot_res(MVR)
1.1000, 312.4898, -121.0000, 191.4898, 0.0000, 191.4898
0.9800, 124.6619, -95.9656, 28.6963, 0.0000, 28.6963
0.7800, -96.6202, -60.7808, -157.4010, 0.0000, -157.4010
0.5800, -206.9895, -33.5960, -240.5855, 0.0000, -240.5855
0.3800, -207.4962, -14.4113, -221.9075, 0.0000, -221.9075
0.1800, -86.3336, -3.2284, -89.5620, 0.0000, -89.5620
</SUBDATA>
}
```

Data QVCurve_Options

Sim_Solution_Options

This subdata section contains solution options that will be used when running QV Curves. See explanation under the CTG_Options object type for more information.

Data SelectByCriteriaSet

SelectByCriteriaSetType

This subdata section contains a list of the display object types which are chosen to be selected. Each line of the section consists of the following:

DisplayObjectType: The object type of the display object.

(FilterName): A filter to apply to these types of objects. This field is optional, but must be given if either of the following fields are given.

(WhichFields): For display objects that can reference different fields, this sets which of those fields it should select (e.g. select only Bus Name Fields). The value may be either ALL or SPECIFIED.

(ListOfFields): If WhichFields is set to SPECIFIED, then a delimited list of fields follows.

Example:

```
<SUBDATA SelectByCriteriaSetType>
DisplayAreaField "" "ALL"
DisplayBus ""
DisplayBusField "Name of Bus Filter" "SPECIFIED" BusName BusPUVolt BusNum
DisplayCircuitBreaker ""
DisplaySubstation ""
DisplaySubstationField "" "SPECIFIED" SubName SubNum BusNomVolt BGLoadMVR
DisplayTransmissionLine ""
DisplayTransmissionLineField "" "ALL"
</SUBDATA>
```

Area

This subdata section contains a list of areas which were chosen to be selected. Each line of the section consists of either the number or the name. When generated automatically by PowerWorld we also include the other identifier as a comment.

Example:

```
<SUBDATA Area>
18 // NEVADA
22 // SANDIEGO
30 // PG AND E
52 // AQUILA
</SUBDATA>
```

Zone

This subdata section contains a list of zones which were chosen to be selected. Each line of the section consists of either the number or the name. When generated automatically by PowerWorld we also include the other identifier as a comment.

Example:

```
<SUBDATA Zone>  
680 // ID SOLUT  
682 // WY NE IN  
</SUBDATA>
```

ScreenLayer

This subdata section contains a list of screen layers which were chosen to be selected. Each line of the section consists of either the name.

Example:

```
<SUBDATA ScreenLayer>  
"Border"  
"Transmission Line Objects"  
</SUBDATA>
```

Data ShapeFileExportDescription

This object uses the same subdata sections as **SelectByCriteriaSet**. The only distinction is that only buses and lines can be exported.

Data StudyMWTransactions

ImportExportBidCurve

This subdata section contains the piecewise linear transactions cost curves for areas involved in a MW transaction. Costs are only for areas that are not on OPF control. Curves must be monotonically increasing. Each line corresponds to a point in the cost curve, and it has two values:

MW: The MW value. Use negative values for imports (purchase) and positive values for exports (sales).

Price: The price in \$/MWh.

Example:

```
<SUBDATA ImportExportBidCurve>
//MW Price[$/MWh]
-20.00 5.00
-10.00 10.00
0.00 15.00
10.00 20.00
20.00 45.00
30.00 70.00
</SUBDATA>
```

Data SuperArea

SuperAreaArea

This subdata section contains a list of areas within each super area. Each line of text contains two values, the area number followed by a participation factor for the area that can be optionally used.

Example:

```
<SUBDATA SuperAreaArea>  
1 48.9  
2 34.2  
25 11.2  
</SUBDATA>
```

Data TSSchedule

SchedPointList

This section stores the schedule time points used in Time Step Simulation. Each line contains seven values:

Date: The date of the point.

Hour: The hour of the point.

Pointtype: An integer specifying the point type.

0 – Numeric

1 – Boolean (Yes/No, Closed/Open)

2 – Text

Numeric Value: The numeric value if point type is Numeric. Otherwise it is just zero.

Boolean Value: The boolean value if point type is Boolean. Otherwise it is just false.

Text Value: The text value if point type is Text. Otherwise it is just an empty string.

Audiofilename: The audio filename associated to the point. If none, it is just an empty string.

Example:

```
<SUBDATA SchedPointList>
//Date Hour PointType NValue BValue TValue AValue
5/8/2006 0 1.00 NO
5/8/2006 6:00:00 AM 0 1.10 NO
5/8/2006 12:00:00 PM 0 1.25 NO
</SUBDATA>
```

Data UserDefinedDataGrid

ColumnInfo

This follows the same convention as the ColumnInfo SUBDATA section described with the [DataGrid](#) objecttype.

Data View

ScreenLayer

This is a list of screen layer names that are hidden in the current view. Each screen layer name is on a separate line of text.

Example:

```
<SUBDATA ScreenLayer>  
//These are hidden screen layers  
"pie layer"  
</SUBDATA>
```

Data Section for Display Auxiliary File

The Data Section of the script file format can contain numerous functions for modifying the data of a power system case. The DATA section begins with a left curly brace and ends with a right curly brace.

DATA DataName (object_type, [list_of_fields], file_type_specifier)

```
{  
data_list_1  
.  
.  
.  
data_list_n  
}
```

Immediately following the DATA keyword, you may optionally include a DataName. Currently, this is not used for anything because there are no script commands that are supported with the *display auxiliary* files.

The first line of the DATA section is referred to as the [Data Argument List](#). The [Data Argument List](#) defines the type and format of the data contained in the following section enclosed in curly braces.

Inside of each DATA section, [data lists](#) are given according to the data definition given in the [Data Argument List](#).

Data Argument List for Display Auxiliary File

The DATA argument list identifies what the information section contains. A left and right parenthesis "(" and ")" mark the beginning and end of the argument list.

The **file_type_specifier** parameter distinguishes the information section as containing custom auxiliary data (as opposed to Simulator's native auxiliary formats), and indicates the format of the data. Currently, the parser recognizes two values for **file_type_specifier**:

(blank) or AUXDEF	Data fields are space delimited
CSV, AUXCSV, or CSVAUX	Data fields are comma delimited

The **object_type** parameter identifies the type of object or data element the information section describes or models. For example, if **object_type** equals DISPLAYBUS, then the data describes DISPLAY BUS objects. Below is a list of many of the object types recognized by Simulator:

CaseInformationMemo	DisplayLineFlowArrow	DisplayTransmissionLineField
ColorMap	DisplayLoad	DisplayZone
Contour	DisplayLoadField	DisplayZoneField
CustomColors	DisplayMultiSectionLine	DisplayZonePie
DisplayArea	DisplayMultiSectionLineField	DocumentLink
DisplayAreaField	DisplayMultiSectionLinePie	DynamicFormatting
DisplayAreaPie	DisplayOwner	Ellipse
DisplayBranchGauge	DisplayOwnerField	Filter
DisplayBranchPie	DisplayOwnerPie	GeoDataViewStyle
DisplayBus	DisplaySeriesCapacitor	GeographyDisplayOptions
DisplayBusField	DisplaySeriesCapacitorField	Group
DisplayBusPie	DisplayShunt	Line
DisplayCircuitBreaker	DisplayShuntField	OnlineField
DisplayDCTransmissionLine	DisplayShuntPie	OnlineLink
DisplayDCTransmissionLineField	DisplaySubstation	Picture
DisplayGen	DisplaySubstationField	PieChartGaugeStyle
DisplayGenericModelField	DisplaySubstationPie	PWFormOptions
DisplayGenField	DisplaySuperArea	Rectangle
DisplayGenPie	DisplaySuperAreaField	Screenlayer
DisplayInjectionGroup	DisplaySuperAreaPie	SelectByCriteriaSet
DisplayInjectionGroupPie	DisplayTransformer	Text
DisplayInterface	DisplayTransformerField	TextBox
DisplayInterfaceField	DisplayTransformerPie	View
DisplayInterfacePie	DisplayTransmissionLine	

The list of object types Simulator's auxiliary file parser can recognize will grow as new applications for the technology are found. Within Simulator, you will always be able to obtain a list of the available

object_types by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields**.

The **list_of_fields** parameter lists the types of values the ensuing records in the data section contain. The order in which the fields are listed in **list_of_fields** dictates the order in which the fields will be read from the file. Simulator currently recognizes over 3,000 different field types, each identified by a specific field name. Because the available fields for an object may grow as new applications are developed, you will always be able to obtain a list of the available **object_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields**. Certainly, only a subset of these fields would be found in a typical custom auxiliary file. In crafting applications to export custom auxiliary files, developers need concern themselves only the fields they need to communicate between their applications and Simulator. A few points of interest regarding the **list_of_fields** are:

- The **list_of_fields** may take up several lines of the text file.
- The **list_of_fields** should be encompassed by braces [].
- When encountering the PowerWorld comment string `'''` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'''` will be ignored as comments.
- Field names must be separated by commas.

Example DATA Argument List

```
DATA (DISPLAYBUS, [BusNom, SOAuxiliaryID, // comment here
SOX, SOY, SOTickness, SOColor, SOUseFillColor, SOFillColor,
// comments allowed here too

// note that blank rows are ignored
SOSize, SOWidth, SOOrientation, SOLevel, SOImmobile,
SLName, SOSStyle, SODashed, // more comments
SOBelongsToGroup])
```

One general note regarding the field names however. Some field names may be augmented with a field location. One example of this is the field `BusNum` when identifying branches. Bus numbers must be used to identify the from and the to end of branches. To keep the number of fields from becoming too large, the same field name is used for both of these values. The from bus number is written as `BusNum:0` and the to bus number is written as `BusNum:1`. Note that the `:0` may be left off of field names.

The **create_if_not_found** field is optional. This specifies whether or not to create a new object if an existing one is not found. If the value is YES, objects will be created. If the value is NO, objects will not be created. If omitted, the default behavior of prompting the user about whether or not to create a non-existing object will continue. If loading and auxiliary file using the **LoadAux** script command, the **create_if_not_found** field for the data section will override the **CreateIfNotFound** field with the script.

Data Key Fields for Display Auxiliary File

Simulator uses certain fields to identify the specific object being described. These fields are called key fields. For example, the key field for DISPLAYBUS objects is BusNum, because a bus can be identified uniquely by its number. The key fields for DISPLAYGEN objects are BusNum and GenID. To properly identify each object, the object's key fields must be present. They can appear in any order in the **list_of_fields** (i.e. they need not be the first fields listed in **list_of_fields**). As long as the key fields are present, Simulator can identify the specific object.

Display objects have an additional key field used for identification because multiple objects can be present on the same one-line diagram that represent the same power system element. This extra key field is SOAuxiliaryID. This is a field that is unique for each type of display object and other key field combination. If there are two display buses that represent bus one in the power system, the SOAuxiliaryID field will be different for both. Simulator will automatically create unique identifiers when these objects are created graphically. They can also be user specified but are forced to be unique. This field does not need to be present when reading in a display auxiliary file, but if it is missing, Simulator assumes that the ID is "1". This field is the only key field identifier for objects that do not link to power system elements such as background lines and pictures, and therefore, should always be included when reading in these objects or the expected results may not be achieved.

By going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields** you will obtain a list of fields available for each object type. In this output, the key fields will appear with asterisks *.

Data List for Display Auxiliary File

After the data argument list is completed, the Data list is given. The data section lists the values of the fields for each object in the order specified in **list_of_fields**. The data section begins with a left curly brace and ends with a right curly brace. A few points of interest regarding the **value_list**:

- The **value_list** may take up several lines of the text file.
- Each new data object must start on its own line of text.
- When encountering the PowerWorld comment string `'//'` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'//'` will be ignored as comments.
- Remember that the right curly brace must appear on its own line at the end of the **data_list**.
- If the **file_type_specifier** is CSV, the values should be separated by commas. Otherwise, separate the field names using spaces.
- Strings can be enclosed in double quotes, but this is not required. You should however always enclose strings that contain spaces (or commas) in quotes. Otherwise, strings containing commas would cause errors for comma-delimited files, and spaces would cause errors for space-delimited formatted files.

Special Data Sections

There are several object types that should be noted here because they can impact the reading of an entire display auxiliary file, overall look of the resulting one-line diagram, or require special input to properly import/export the object.

GeographyDisplayOptions

Most objects supported in the display auxiliary file have coordinates that can be specified in the appropriate data sections. What these coordinates specify can be controlled by the GEOGRAPHYDISPLAYOPTIONS object. This object has only two fields available: MapProjection and ShowLonLat. There are three possible settings for MapProjection: "x,y", "Simple Conic", and "Mercator". The choice of projection will determine how the x,y values for display objects are interpreted. ShowLonLat can be either "YES" or "NO". If ShowLonLat is "YES", the setting specified for the MapProjection will be the longitude,latitude projection used when reading/writing the object x,y values. If ShowLonLat is "NO", the x,y values will always be interpreted as x,y regardless of the MapProjection setting. This object should be placed in the display auxiliary file before any other objects containing coordinates are read. If this object is not included in the auxiliary file, the coordinates will be interpreted based on the current settings of map projection and whether or not coordinates are showing longitude,latitude.

Picture

PICTURE objects represent background images that cannot be stored in a text file format. To properly include a PICTURE object in a display auxiliary file, the file containing the image must be saved and read along with the auxiliary file. The FileName field indicates the name and location of the image file. If the image file cannot be found when reading in a display auxiliary file and attempting to create a new object, no PICTURE object will be created. If attempting to update an existing object and the image file cannot be found, the object will not be updated with a new image, but the FileName field will be updated with the specified file name.

PWFormOptions

One-line display options that affect the current display settings can be changed by using the PWFORMOPTIONS object. Usually, this object specifies named sets of options that can be selected and used to change the various one-line display options through the GUI. By including a specially named object, the current options can be changed through a display auxiliary file. PWFORMOPTIONS are named using the OOName field. Setting this field to "THESE_OPTIONS_ARE_APPLIED_TO_THE_CURRENT_DISPLAY" will apply the specified set of options to the current one-line when the file is read. When saving the entire one-line to a display auxiliary file, a PWFORMOPTIONS object with this name is added to the file by default.

View

Different views can be specified in the display auxiliary file using the VIEW object. Usually, this object is used to specify named sets of options used to select and change the view through the GUI. By including a specially named object, the current view can be changed through a display auxiliary file. VIEW objects are named using the ViewName field. Setting this field to "THIS_VIEW_IS_APPLIED_TO_THE_CURRENT_DISPLAY" will apply the specified set of view options to the current one-line when the file is read. When saving the entire one-line to a display auxiliary file, a VIEW object with this name is added to the file by default.

Data Subdata Sections for Display Auxiliary File

The format described in the [Data Section](#) works well for most kinds of data in Simulator. It does not work as well however for data that stores a list of objects. For example, a contingency stores some information about itself (such as its name), and then a list of contingency elements, and possible a list of limit violations as well. For data such as this, Simulator allows `<subData>`, `</subData>` tags that store lists of information about a particular object. This formatting looks like the following:

DATA (object_type, [list_of_fields], file_type_specifier)

```
{
value_list_1
  <SUBDATA subobject_type1>
    precise format describing an object_type1
    precise format describing an object_type1
    .
    .
    .
  </SUBDATA>
  <SUBDATA subobject_type2>
    precise format describing an object_type2
    precise format describing an object_type2
    .
    .
    .
  </SUBDATA>
value_list_2
.
.
.
value_list_n
}
```

Note that the information contained inside the `<SubData>`, `</SubData>` tags may not be flexibly defined. It must be written in a precisely defined order that will be documented for each SubData type.

Data Common Sections

ColorMap

Same format as in *data auxiliary* files.

CustomColors

Same format as in *data auxiliary* files.

DynamicFormatting

Same format as in *data auxiliary* files.

Filter

Same format as in *data auxiliary* files.

GeoDataViewStyle

Same format as in *data auxiliary* files.

PWFormOptions

Same format as in *data auxiliary* files.

SelectByCriteriaSet

Same format as in *data auxiliary* files.

UserDefinedDataGrid

Same format as in *data auxiliary* files.

Data DisplayLine

DisplayDCTramissionLine**DisplayInterface****DisplayMultiSectionLine****DisplaySeriesCapacitor****DisplayTransformer****DisplayTransmissionLine****Line**

This is a list of points defining the graphical line used to represent the object. Each set of coordinates can be enclosed in square brackets, [], or the brackets can be eliminated. The brackets will be included when Simulator generates an auxiliary file. The individual coordinates are separated by the specified delimiter, either a space or a comma, and if the brackets are included, the same delimiter should be used to separate sets of coordinates. The list of points is in a somewhat free form and sets of coordinates can span multiple lines. Each point should either be in x,y coordinates or longitude,latitude coordinates. Which coordinates should be used depends on the current option settings for map projection and whether or not coordinates should be shown in longitude,latitude. If the display auxiliary file is automatically generated by Simulator, a comment will be included in the subdata section indicating the coordinate system in use during file creation.

Example using brackets and a comma delimiter:

```
<SUBDATA Line>
//Coordinates are x,y
[14.00000000, 63.00000000], [14.00000000, 60.00000000],
[20.00000000, 45.00000000], [20.00000000, 42.00000000]
</SUBDATA>
```

Example with no brackets and a space delimiter:

```
<SUBDATA Line>
//Coordinates are x,y
14.00000000 63.00000000 14.00000000 60.00000000
20.00000000 45.00000000 20.00000000 42.00000000
</SUBDATA>
```

Data PieChartGaugeStyle

ColorMap

This is a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values:

Percentage: This is the percentage at which the following scalar and color should be applied.

Scalar: A scalar (multiplier) on the size of the pie chart/gauge.

Color: A color for the pie chart/gauge.

Example:

```
<SUBDATA ColorMap>  
//Percentage Scalar Color  
85.0000 1.5000 33023  
100.0000 2.0000 255  
</SUBDATA>
```

Data View

ScreenLayer

This is a list of screen layer names that are hidden in the current view. Each screen layer name is on a separate line of text.

Example:

```
<SUBDATA ScreenLayer>  
//These are hidden screen layers  
"pie layer"  
</SUBDATA>
```

Chapter 14: PowerWorld Simulator Project Files

PowerWorld Project Files provide the user an easy way consolidate and manage files from any application, including PowerWorld binary and display files.

This chapter covers the following:

- Overview of PowerWorld Simulator project files
- Creating a New Project File
- Opening a Project File

Overview of PowerWorld Simulator Project Files

Performing a simulation using PowerWorld Simulator sometimes requires using a number of different files. In addition to the case file that stores the model of the system, there may be one or more oneline diagrams depicting various regions of the system. These oneline diagrams might feature [document links](#) that connect to files that were created using other applications, as well as [oneline links](#), which open other oneline diagrams when they are clicked. For these links to function properly, the documents and oneline displays to which they connect must be available. A simulation might also employ a template file that is used to load in a predefined set of solution, environment, and display options. Furthermore, the simulation might make use of data stored in [auxiliary data files](#) to supplement the data stored in the case file. Finally, a simulation might utilize a [script file](#) to perform some automated sequence of tasks, or even to display a movie of system conditions. Having to deal with so many files may make it difficult to transfer the case to another computer, or to share the model with a colleague. PowerWorld Simulator Project files provide a solution.

PowerWorld Simulator Projects have the extension *.pwp. A project serves as a container for all the files that might comprise a simulation, including the case file, one or more oneline diagrams, a script file, one or more auxiliary data files, and a case template, as well as any other support files you may wish to include. A project is actually a compressed file archive that is compatible with the widely available PKZip and WinZip file compression utilities. It is strongly recommended, however, that you work with project files strictly within the PowerWorld Simulator environment, as Simulator can automatically perform the file compression and extraction functions and process the contents of the included files in a single step. (Note: Simulator performs file compression and extraction using software components available from <http://www.cdrom.com/pub/infozip/> and <http://www.geocities.com/SiliconValley/Orchard/8607/main.html>).

PowerWorld Project Initialization Script

Included in every project file is an initialization script file called OPENPWB.SCP. This is a special type of PowerWorld script file that is used to unload the contents of the project that are used in the simulation. This file must begin with the keyword "INITIALIZATION" and terminate with the keyword "END". Between these two lines, OPENPWB.SCP identifies the name of the case, any oneline diagrams that should be opened immediately, a template file (if any), a script file (if any), and one or more auxiliary data files (if any) to open and read into memory after extracting the files. The file might also contain the keyword "AUTOSTART", indicating that the simulation of the system described by the project file should commence immediately after it is read into memory. The OPENPWB.SCP is generated for you automatically when you save a new project file. Here is an example that loads the case B7FLAT.PWB (in PowerWorld binary format) along with oneline diagram B7FLAT.PWD, the template B7FLAT.PWT, the script B7FLAT.SCP, and the auxiliary file B7FLAT.AUX, and then starts the simulation immediately after all contents are extracted and read:

```
INITIALIZATION
CASE PWB B7FLAT.PWB
ONELINE B7FLAT.PWD
TEMPLATE B7FLAT.PWT
SCRIPT B7FLAT.SCP
DATAFILE B7FLAT.AUX
AUTOSTART
END
```

Again, all projects must contain the file OPENPWB.SCP. If a project does not contain this file, an error message will be shown. Simulator automatically includes an OPENPWB.SCP file with every project it creates. Therefore, unless you try to create a project file outside of Simulator, you will not have to worry about this requirement.

Associating Project Files with Simulator

PowerWorld Simulator can automatically open and load the contents of a project if the proper file associations have been made. When Simulator is installed on a computer, it registers the file extension *.pwp in the Windows registry. Then, whenever you double-click on an icon for a project file or download a project file from a web site, Simulator will start, extract the contents of the project file, and read them into memory. The power system will then be displayed exactly as the author of the project file intended it to be displayed.

If the file association is somehow destroyed (i.e. Windows no longer recognizes that *.pwp files should be associated with and opened by Simulator), you can re-establish the association manually using the following procedure:

- Windows 95/98/NT
 - o Open Windows Explorer.
 - o Select View > Folder Options from Windows Explorer's main menu.
 - o Switch to the tab labeled "File Types."
 - o Click the button labeled "New Type."
 - o For "Description of Type," specify "PowerWorld Project."
 - o For "Associated Extension," specify "PWP".
 - o For "ContentType (MIME)," specify "application/powerworld_project."
 - o For "Default Extension," specify ".PWP".
 - o Under "Actions," click the button labeled "New."
 - o Specify the "Action" as "open."
 - o Specify the "Application used to perform action" as the path to the Simulator executable file, followed by a space and the string "%1" (including the quotation marks). For example, if the path of the PowerWorld Simulator executable is "c:\program files\PowerWorld\pwrworld.exe," then you should specify the following string:
c:\program files\PowerWorld\pwrworld.exe "%1"
 - o Click OK to close the Action dialog.
 - o Click OK to complete the definition of the file association.
- Windows XP
 - o Open Windows Explorer
 - o Select Tools > Folder Options from Windows Explorer's main menu.
 - o Switch to the tab labeled "File Types"
 - o Click the button labeled New
 - o For "File Extension", enter "PWP", then click OK.
 - o Find the PWP entry in the Registered file types list, then click on it.
 - o Next, with the PWP entry selected, click on Advanced.
 - o In the box to the left of the "Change Icon..." button, type in "PowerWorld Project"
 - o Click on the "New..." button on the right-hand side of this dialog, which will pop up a "New Action" dialog
 - o In the New Action dialog, click "Browse..."
 - o Navigate to and select the PowerWorld program binary (pwrworld.exe); the default location is c:\Program Files\PowerWorld\pwrworld.exe
 - o In the New Action dialog, enter "Open the project" in the field labeled "Action"
 - o Click OK in the New Action dialog and in the Edit File Type dialog
 - o Click Close when you are back to the main File Types dialog.

The *.pwp file type will now be recognized on your system as being associated with PowerWorld Simulator. Again, you will need to perform this procedure only if the association somehow gets removed.

Creating a New Project File

Creating a new project file in Simulator is fairly simple. First, make sure that the case for which you want to create the project is open in Simulator, as well as any oneline diagrams that you wish to open automatically whenever the user opens the project. Then select **Save As Project** from the [Application Button](#). This opens the [Create Project Dialog](#), which you can use to specify the contents of the project file. When you click OK on the Create Project Dialog, the files you identified will be compressed into a single file, along with the [project initialization script](#) that tells Simulator how it should import the contents of the project.

Create Project Dialog

The Create Project Dialog enables you to create a new [PowerWorld Simulator Project](#) by specifying the files that should be included in the project. To open the Create Project Dialog, select **Save As Project** from the [Application Button](#). When it opens, the dialog will automatically include the case file, all open oneline diagrams, all documents and onelines linked to the display by [document links](#) and [oneline links](#), the current case template file, and the current script in its list of files to add to the project. You can then use this dialog to add files to or delete files from the list of project contents.

The Create Project Dialog features the following controls:

Case

Identifies the name of the case file that is currently open in Simulator. The case file may be in PowerWorld binary format, in PTI Version 23 - 29 .raw format, or in GE PSLF .epc format. You cannot change this field, because it must be set to whatever case is currently open.

Open these oneline diagrams automatically

Lists the names of the oneline diagrams that will automatically open whenever a user opens the project file. This is also a read-only control and lists the names of all oneline diagrams that are currently open in Simulator. Therefore, before you start to create a project using the Create Project dialog, make sure that only those oneline diagrams you wish to open automatically with the project are currently open in Simulator.

Use this script file

Identifies a [script file](#) that should be opened automatically when the project is open. Scripts are text files that contain commands in [PowerWorld's scripting language](#) that Simulator interprets to perform a predefined set of tasks during the simulation. If a script file is in use when you try to create the project, its name will automatically appear in this text box. To specify a different script file, either type its full path in the text box, or press the adjacent *Browse* button to search for it on your machine.

Load data from auxiliary files

Lists the [auxiliary data files](#) that should be automatically read when the project is open. Auxiliary data files are text files that contain data that supplement or supersede the information in the case file. You may include as many automatically loading auxiliary data files in the project as you like. To add an auxiliary file to the project contents, click the adjacent Add button and locate the file on your machine. To remove an auxiliary file from the list, select it and click the adjacent Remove button.

Also include these files

Lists any other files you may wish to include in the project. Use this control to include supplementary documents, pictures, or even other PowerWorld project files, in the project. To add a file to the contents, click Add a File. To remove a file from the list, select it and press the adjacent Remove button.

Automatically start the simulation after opening the project

Check this button to cause Simulator to start the simulation immediately after opening the project. This saves the user who opens the project the step of switching to run mode and clicking the play button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. This option is particularly effective, for example, if you want to display a "movie" of system conditions when the project is open. By including a script file in the project that tells Simulator how to modify the system and view over time, you create a project that, when opened or downloaded, animates the scenario you are trying to model.

Save this project as

Identifies the name of the project that should be saved. By default, the project name is the same as that of the case except that it has the extension "pwp". To specify a different name, either type it directly in the text box, or click the adjacent Browse button and specify the full path.

When you have finished specifying the contents of the project file, click the **OK, save the project** button. If you wish to cancel the operation without creating the project, click **Cancel**.

Opening an Existing Project

To open an existing [PowerWorld Simulator Project file](#), select **Open Project** from the [Application Button](#), and select the name of the project you wish to open. Alternatively, you may select **Open Case** from the [Application Button](#), change the *Files of Type* setting to "PowerWorld Project (*.pwp)," and select the project you wish to open. Simulator will extract the contents of the project file into the project file's folder and input the information it needs to display the case, oneline diagrams, and other associated files.

Chapter 15: Tutorials

This chapter contains tutorials on building a new case in Simulator and starting from an existing case in Simulator. They were originally created for use from the on-line help, although they can be used in text format as reproduced here.

- Creating a New Case
- Starting with an Existing Case
- OPF
- Contingency Analysis

Tutorial: Creating a New Case Page 1 of 13

This procedure describes how to create a simple power system model using PowerWorld Simulator. This procedure was developed for use with version 13 and later of the package. If you have an earlier version, please contact PowerWorld Corporation at info@powerworld.com for information on upgrading, or visit the website at <http://www.powerworld.com>.



To begin, double-click on the PowerWorld Simulator icon.  This starts Simulator. Simulator is used to create new cases, modify existing cases, and (of course) simulate power systems. In this example, we will build a new case from scratch. To create a case from an existing power flow case instead, please see [Creating a Case from an Existing Power Flow File](#).



To create a new case, select **New Case** from PowerWorld icon  in the upper left corner of the program. The screen background will turn white, the default background color for new PowerWorld oneline diagram. Oneline diagrams are used in power system analysis to represent the actual three-phase power system using a single line to represent each three-phase device.

Tutorial: Inserting a Bus Page 2 of 13

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices are connected together. In building a power system model using Simulator, you will draw buses onto the oneline diagram, attach devices such as generators and loads to the buses, and connect different buses together with transmission lines and transformers.

To insert a bus:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This prepares Simulator to insert a new bus.
- Left-click on the oneline background at the location where you want to place the new bus. This invokes the [Bus Option Dialog](#) (pictured below), which is used to specify the name, orientation, shape, size, width, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to the bus .

Bus Options

Insert new bus in data model

Bus Number: 1 Find By Number Find ...

Bus Name: One Find By Name

Nominal Voltage: 138.00 kV

Labels ...

	Number	Name
Area	1	1
Zone	1	1
Owner	1	1
Substation		

Bus Information **Display** Attached Devices Custom

Orientation: Right Up Left Down

Shape: Rectangle Ellipse

Size: 5.00 Width: 0.200 Scale Width with Size:

Link to New Bus

OK Save Cancel

Bus Options

Insert new bus in data model

Bus Number: 1

Bus Name: One

Nominal Voltage: 138.00 kV

Labels ...

	Change	Number	Name
Area	<input type="button" value="Change"/>	1	1
Zone	<input type="button" value="Change"/>	1	1
Owner	<input type="button" value="Change"/>	1	1
Substation	<input type="button" value="Change"/>		

Bus Information

Bus Voltage

Voltage (p.u.): 1.00 System Slack Bus

Angle (degrees): 0.00

- The *Bus Number* field automatically displays '1'. Simulator requires that each bus have a unique number. For convenience, accept the default value. Historically, the *Bus Name* field was limited to eight characters. That limitation is no longer imposed in Simulator. Many users still choose to limit the bus name to eight characters by convention and for ease in converting cases to other formats (such as .epc or .raw). For this example, enter '**One**'.
- Next, check the *System Slack Bus* field, which is located in the Bus Voltage portion of the Bus Information Tab. The Slack Bus is a modeling construct that ensures that the power system has enough generation to meet the load. In other words, the slack bus "picks up the slack" caused by system losses or unbalanced generation and load.
- Click **OK** on the Bus Option Dialog to finish creating the bus and to close the dialog. After the dialog box closes, the new bus appears on the oneline at the location you specified.

Tutorial: Inserting a Generator Page 3 of 13

Next we are going to attach a generator to the bus. Generators may be inserted in a manner similar to inserting a bus:

- Select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click the bus on the oneline diagram to which you want to attach the generator (for this example, click on the slack bus – bus *One*.) The [Generator Option Dialog](#) (pictured below) will automatically open. The dialog is used to specify the new generator's unit identifier, display size, orientation, MW output and limits, reactive power limits, set point voltage, and cost model.

Generator Options

Bus Number: 1
 Bus Name: One
 ID: 1
 Area Name: 1
 Labels ...
 Find By Number
 Find By Name
 Find ...
 Status: Open Closed
 Fuel Type: Unknown
 Unit Type: UN (Unknown)

Display Information | **Power and Voltage Control** | Costs | Fault Parameters | Owner, Area, Zone, Sub | Custom

Power Control

MW Output: 413
 Min. MW Output: 0.000
 Max. MW Output: 1000.000
 Available for AGC
 Enforce MW Limits
 Part. Factor: 10.00
 MW Ramp Limit: 20.0

Voltage Control

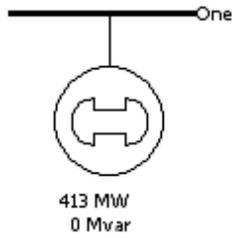
Mvar Output: 0.000
 Min Mvars: -9900.000
 Max Mvars: 9900.000
 Available for AVR
 Use Capability Curve
 Regulated Bus Number: 1
 SetPoint Voltage: 1.0000
 Remote Reg %: 100.0

MW									
Min Mvar									
Max Mvar									

OK Save Cancel Help

- Every generator must have a *MW Output* specified when the generator is inserted. Make sure the *MW and Voltage Control* tab is selected. Enter '413' in the *MW Output* Field. Note: the *MW Output* specified for a generator connected to the system slack bus is arbitrary because the generator's true output depends on system load and losses.
- Select the *Display Information* tab. The *Orientation* field is used to specify the direction the generator will extend from the bus. The *Anchored* checkbox forces the generator to move with its specified bus when repositioning the bus on the oneline.

- Click **OK** on the Generator Option Dialog to accept the default values for all other fields. After the dialog box closes, the new generator will appear on the oneline attached to the previously selected bus. The oneline diagram should resemble the image shown below.



Tutorial: Saving the Case Page 4 of 13

To save the work that we have done so far, select **Save Case** from the [Application Button](#), or click on the *Save Case*  button. Before the case is saved, Simulator validates the case to make sure that it does not contain any errors. Results from this validation are displayed in the Message Log display, usually shown in the lower right-hand corner of the display. If the log is not visible, click the **Log** on the Log group under **Tools**. Since we have not yet named the case, the **Save As** dialog is displayed. Enter a file name and select **OK**. By default the case is saved using the [PowerWorld Binary format](#) (*.pwb). When saving the case in the future, you will not have to reenter its name. Simulator also asks you to supply a name for saving the oneline diagram we have been drawing. The oneline diagram files have a default extension of *.pwd, which identifies them as PowerWorld Display files. Supply the same name as you gave to the case. Note that, because the case and the oneline are stored in separate files, multiple onelines can be assigned to the same case, and the same oneline can be used by many cases.

Tutorial: Entering a Second Bus with Load Page 5 of 13

To enter the second bus:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on the oneline diagram somewhere to the right of the first bus. In the Bus Options Dialog (pictured below) leave the bus number at the default value of **2**, and enter the name '**Two**' in the *Bus Name* field.
- We will model a 200 MW, 100 Mvar load at the bus. Select the *Attached Devices* tab. Under the *Load Summary Information* heading enter '**200**' in the *Base MW* field and '**100**' in the *Base Mvar* field.
- Click **OK** to accept all other default values, close the Bus Options Dialog, and insert the bus.

The screenshot shows the 'Bus Options' dialog box with the following fields and values:

- Bus Number:** 2
- Bus Name:** Two
- Nominal Voltage:** 138.0 kV
- Labels:** no labels
- Area:** 1
- Zone:** 1
- Owner:** 1
- Substation:** (empty)
- Attached Devices Tab:**
 - Load Summary Information:**
 - Base MW: 200.0
 - Base Mvar: 100.0
 - Shunt Admittance:**
 - G (MW): 0.0
 - B (Mvar): 0.0

At this point, the oneline diagram does not show the load at bus 2, even though it is represented in the power system model (you can confirm this by right-clicking on bus 2, selecting *Bus Information Dialog* from the resulting local menu, and inspecting the *Load Summary Information* fields again).

To draw the load on the oneline diagram:

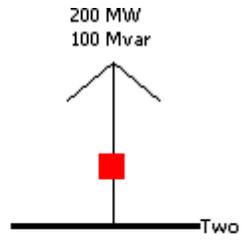
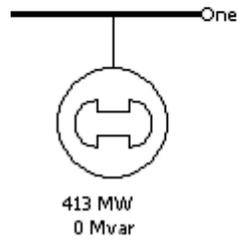
- Select **Network > Load** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click in the center of this bus. The Load Options Dialog box (pictured below) automatically opens. The *Constant Power MW* and *Mvar* fields confirm that the load is 200 MW and 100 Mvar. In addition to constant power loads, Simulator also allows the modeling of voltage dependent loads.

- Select **Up** in the *Orientation* field under the *Load Information* tab to make the load point up. Verify that the anchored box is checked to force the load to move with the selected bus.
- Click **OK** to accept the default values for all remaining fields, close the Load Options dialog, and insert the load. A circuit breaker symbol is automatically included with each load.

To move objects on the oneline:

- Left-click on the desired object. Drag and drop the object to the new location by holding the left mouse button down while moving the mouse. Note: you can also move all objects on the oneline simultaneously by left-clicking on the diagram (not on a specific object) then dragging and dropping in the desired location.
- To move bus 2, left click on bus 2 (not on the attached load). Drag the bus to a new location. Note that the load moves with the bus because it is [anchored](#). You can change the location of attached devices connected to a bus, such as generators and loads, by the same procedure.

The oneline diagram should now resemble the image shown below.



Tutorial: Inserting a Transmission Line Page 6 of 13

Transmission lines are used to connect buses together. To insert a transmission line:

- Select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click at the point where you want the new line to originate. This point is usually located on one of the proposed line's terminal buses. For this example, originate the line at bus One.
- Transmission lines and transformers are drawn as a series of line segments. Without holding down the mouse button, drag the mouse up. Notice that a line segment connected to the point of origin will follow your mouse movements. To terminate a line segment, click the left mouse button. Each time you click the mouse to terminate a line segment, a new vertex is defined for the line. To draw the next line segment, move the mouse to the desired location of the next vertex. Note: the vertices may later be moved or deleted to reshape the line. To create curved lines, hold the left mouse button down while dragging.
- To terminate the final line segment and conclude drawing the line, double click the left mouse button at the desired termination point (bus Two for this example). The termination point is usually the transmission line's other terminal bus.
- The [Transmission Line/Transformer Dialog](#) automatically appears (shown below). The dialog should already contain a **1** in the *From Bus Number* field and a **2** in the *To Bus Number* Field. If not, you probably did not have the cursor directly on the bus when you were drawing the line. If this is the case, simply enter the correct bus numbers in the corresponding fields.

Transmission Line/Transformer Options

From Bus: 1, To Bus: 2, Circuit: 1

Number: 1, Name: One, Area Name: 1 (1), Nominal kV: 138.0

Labels ...

Find By Numbers, Find By Names, Find ...

From End Metered

Default Owner (Same as From Bus)

Labels ...

Fault Info, Owner, Area, Zone, Sub, Custom

Display, Parameters, Transformer Control, Series Capacitor

Status: Open, Closed

Length (mi): 0.00

Calculate Impedances >

Per Unit Impedance Parameters:

Series Resistance (R): 0.02

Series Reactance (X): 0.08

Shunt Charging (B): 0.1

Shunt Conductance (G): 0.0000

Has Line Shunts

Line Shunts

MVA Limits:

Limit A	1000
Limit B	0.000
Limit C	0.000
Limit D	0.000
Limit E	0.000
Limit F	0.000
Limit G	0.000
Limit H	0.000

Convert Line to Transformer

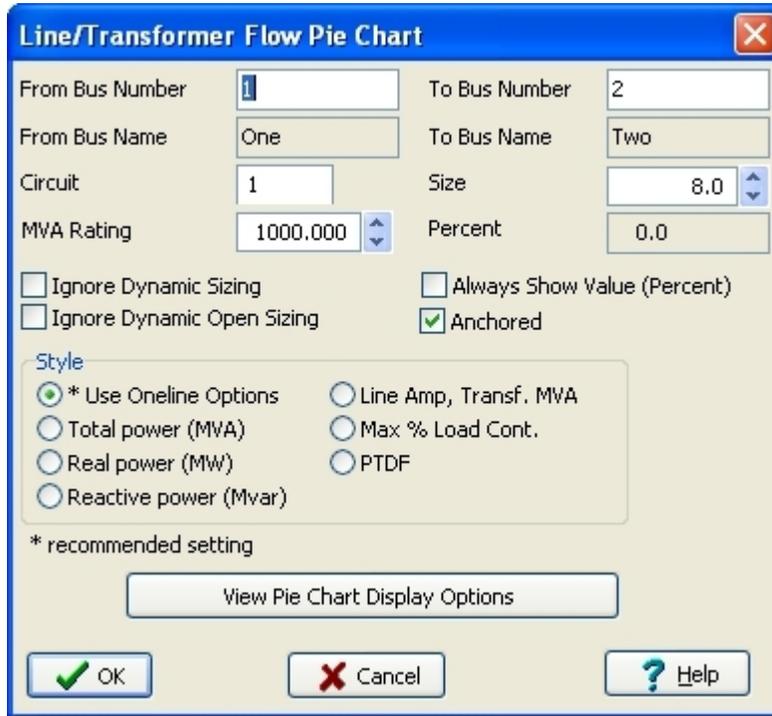
OK, Save, Cancel, Help

- The *Series Resistance*, *Series Reactance*, and *Shunt Charging* fields are used to enter the per unit parameters associated with the line. The *Shunt Charging* field contains the total per unit charging capacitance for the line. Enter **0.02** in the *Resistance* field, **0.08** in the *Reactance* field and **0.1** in the *Shunt Charging* field.
- The *Limit (MVA)* fields contain the MVA ratings for the line; enter a value of 1000 in the *Limit A (MVA)* field.
- Click **OK** to accept all remaining field default values, close the Transmission Line/Transformer Dialog, and insert the new line.

By default, the transmission line is anchored to both terminal buses. If you try to move bus 2, the transmission line should move with it.

Tutorial: Inserting a Line Flow Pie Chart on a Transmission Line Page 7 of 13

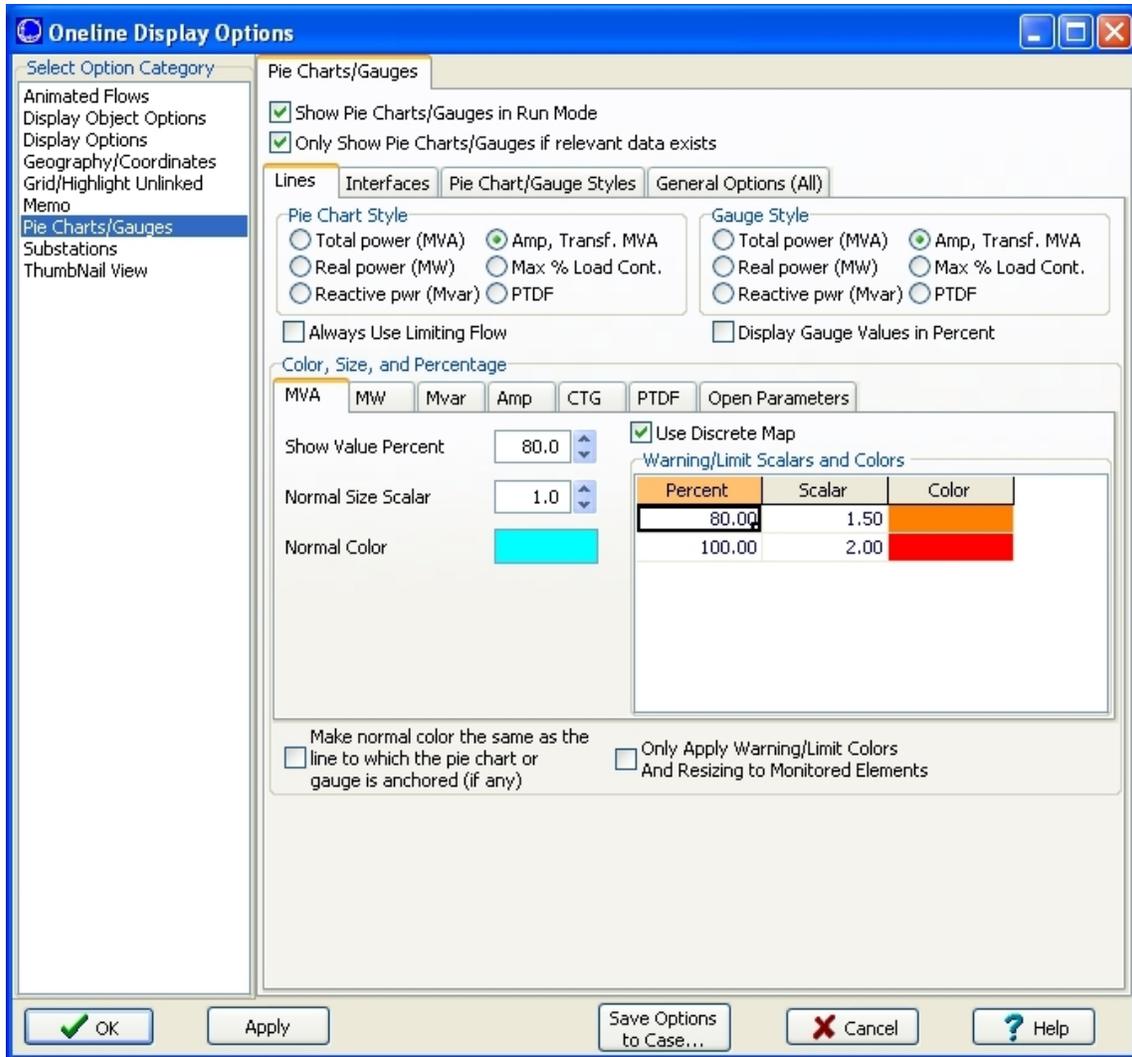
When the line is drawn it automatically it has a line flow pie chart included. You can include additional line flow pie charts by selecting **Pies/Gauges > Line Flow Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then clicking near a line. The Line/Transformer Flow Pie Chart dialog box appears (shown below). Make sure that the From Bus and To Bus have the correct numbers, that the MVA rating is correct, and that Anchored is checked. You may change the size of the pie chart by typing in a value or using the arrows.



The dialog box titled "Line/Transformer Flow Pie Chart" contains the following fields and options:

- From Bus Number: 1
- To Bus Number: 2
- From Bus Name: One
- To Bus Name: Two
- Circuit: 1
- Size: 8.0
- MVA Rating: 1000.000
- Percent: 0.0
- Ignore Dynamic Sizing
- Always Show Value (Percent)
- Ignore Dynamic Open Sizing
- Anchored
- Style section:
 - * Use Oneline Options
 - Total power (MVA)
 - Real power (MW)
 - Reactive power (Mvar)
 - Line Amp, Transf. MVA
 - Max % Load Cont.
 - PTDF
- * recommended setting
- View Pie Chart Display Options button
- OK, Cancel, and Help buttons at the bottom.

Color and behavior of pie charts are set in the *Oneline Display Options* display; right-click anywhere in the background of the oneline, select *Oneline Display Options* from the pop-up menu, then select the *Pie Charts* tab (pictured below).

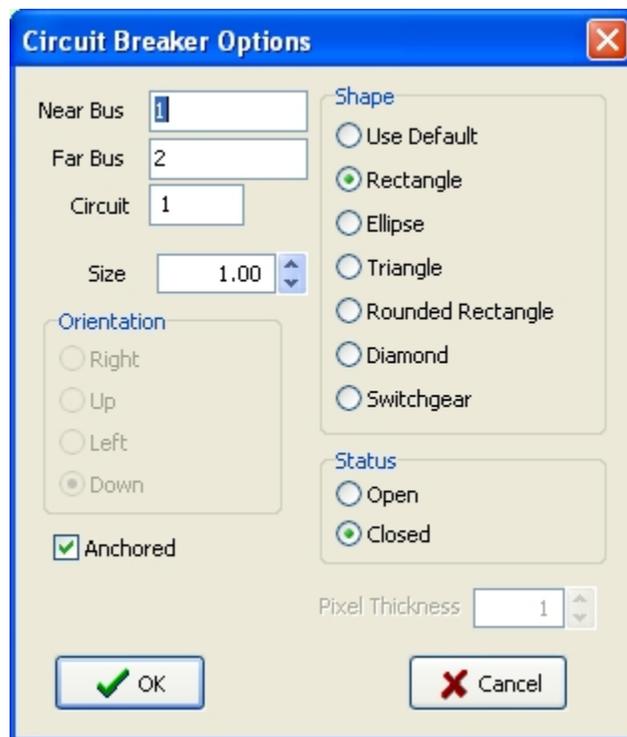


Tutorial: Inserting Circuit Breakers Page 8 of 13

Circuit breakers are used to control the status of the line. (If the line already has circuit breakers at each end, then Simulator has been instructed to insert circuit breakers automatically. You can configure this option from the [Default Drawing Options Dialog](#)).

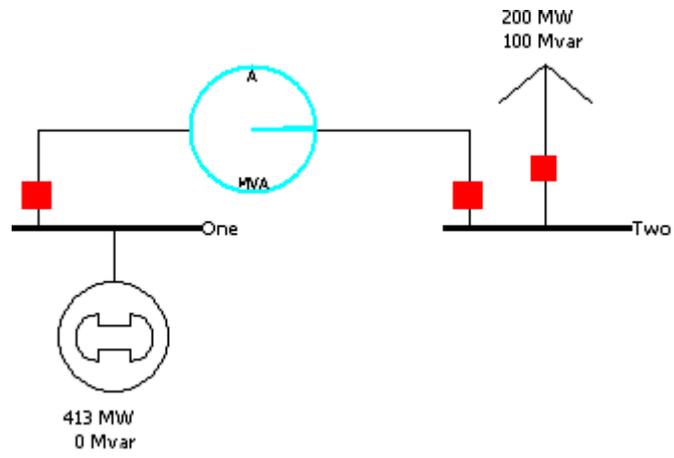
To insert a Circuit Breaker:

- Click somewhere on the line near bus One then select **Indication > Circuit Breaker** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, then click on the line near bus One. You should immediately see the Circuit Breaker Options dialog (shown below) with the From Bus and To Bus fields correctly set to '1' and '2'. If they are '0', enter the correct value. Set the Size field to '1' (you can either enter a 1, or use the spin arrows to change the value).
- Click **OK** to insert the circuit breaker.



In Simulator, the location of the circuit breaker does not matter, because changing the status of the circuit breaker changes the status of the entire line. However, since most transmission lines have circuit breakers at each end, we will also place a circuit breaker near bus 2. To accomplish this, repeat the above process near bus 2.

Save your case. Your oneline should now look similar to the image below.



Tutorial: Inserting a Transformer Page 9 of 13

To insert a transformer we first need to insert a bus at a different voltage level. Insert a new bus in the bottom of the oneline, named **Three**, and enter **69** KV for its nominal voltage in the Bus Options dialog box.

To insert a transformer between buses 2 and 3:

- Either go to **Network > Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on bus 2, and then draw a line to bus 3 as you did for a transmission line. The [Transmission Line/Transformer Dialog](#) automatically appears.

- On the **Parameters** tab, enter **0.02** for *Series Resistance*, **0.08** for *Series Reactance*, **0.1** for *Shunt Charging* and **1000** for *Limit A (MVA)*.
- Select the **Transformer Control** tab. Note that the Off-nominal Turns Ratio displays **1.000**. The true transformer turns ratio does not need to be specified as it is automatically determined by the

ratio of nominal voltages between the *From Bus* and *To Bus*. The *Off-nominal Turns Ratio* is used to adjust the transformer tap setting on per-unit values of bus voltages as referenced to their respective base values for per-unit calculation.

- Click **OK** to accept the default values, close the dialog and insert the transformer.
- Repeat this procedure to add a transformer between buses 1 and 3.

Add a 400 MW, 200 Mvar load to bus 3:

- Right-click on bus 3 and select **Bus Information Dialog**. Select the **Attached Devices** tab. Click **Add or Edit Bus Load**.
- Under *Constant Power* enter **400** in the *MW Value* field. Enter **200** in *Mvar Value* field. Click **OK**.
- Note that *Base MW* and *Base Mvar* display the respective values. Click **OK**.

The load is now attached to the bus even though it is not displayed on the oneline. To display the load as an object on the oneline, you can either use the **Auto Insert** feature or follow the procedure utilized earlier. To **Auto Insert** the load:

- Select **Auto Insert > Loads...** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. The *Automatic Insertion of Loads* dialog opens automatically.
- Click **OK** to accept the default values and insert the load object on the oneline.

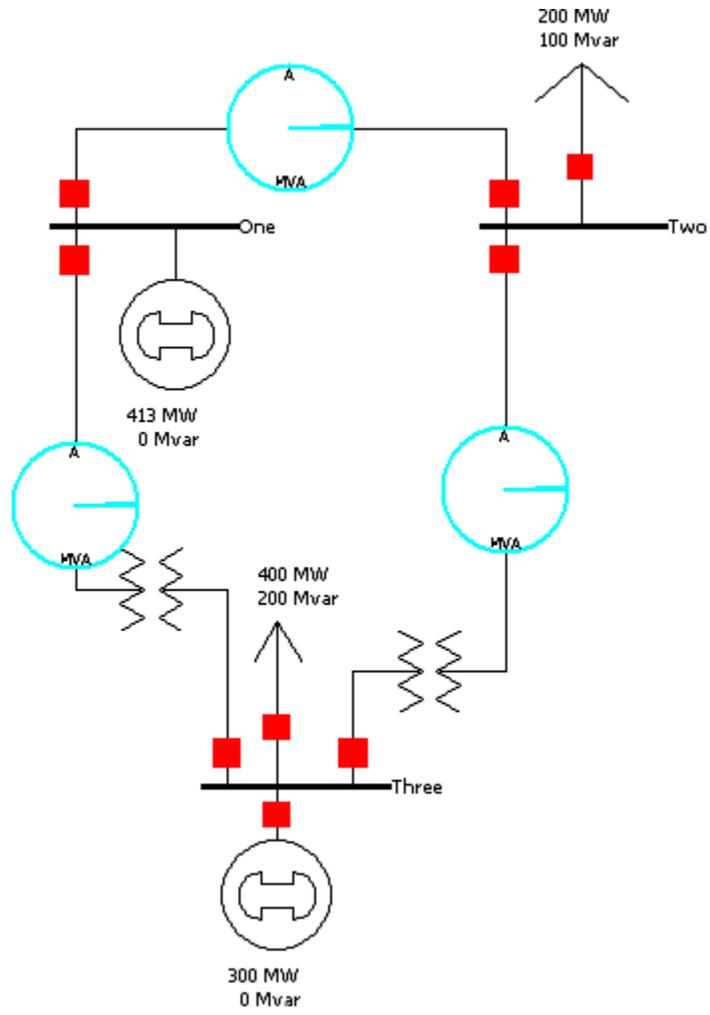
Note that Lines, Loads, Interfaces, Generators and Switched Shunts can all be inserted as objects on a oneline using the **Auto Insert** tool if a record already exists for the device.

- Left-click and drag the load to the desired location on bus 3. Note: you can resize the bus object on the oneline by left-clicking on the bus then dragging either end-point vertex to the desired bus size.
- Right-click on the load and select **Load Information Dialog**. You can change the load orientation and verify all load parameters in this dialog.
- Click **OK**.

Next we are going to attach a 300 MW generator to the bus 3:

- Select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click on bus 3. The [Generator Option Dialog](#) opens.
- Select the *Power and Voltage Control* tab. Enter '**300**' in the *MW Output* Field.
- Click **OK** on the Generator Option Dialog to accept the default values for all other fields. After the dialog box closes, the new generator appears on the oneline attached to bus 3.

Your oneline should now resemble the image shown below.



Tutorial: Inserting a Switched Shunt Page 10 of 13

Switched shunts usually consist of either capacitors to supply reactive power (in MVAR) to the system, or reactors to absorb reactive power. The switched shunts are represented by a number of blocks of admittance that can be switched in a number of discrete steps. If at least one block is in service, the shunt is said to be online. The shunt's corresponding circuit breaker is used to determine and / or toggle the switched shunt's status.

- To insert a switched shunt at bus 3, select **Network > Switched Shunt** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on or near bus 3. The *Switched Shunt Options* dialog box appears (pictured below).
- Verify that the bus number is 3; if it is not, change it.
- Enter **10** for the *Nominal Mvar*.
- Click **OK** to accept the default values of the remaining fields, close the dialog, and insert the switched shunt.

Switched Shunt Options

Bus Number: 3

Bus Name: Three

Shunt ID: 1

	Number	Name
Area	1	1
Zone	1	1
Substation		

Display Parameters Control Parameters Fault Parameters Custom

Nominal Mvar:

Control Mode: Fixed Discrete Continuous Bus Shunt (Fixed)

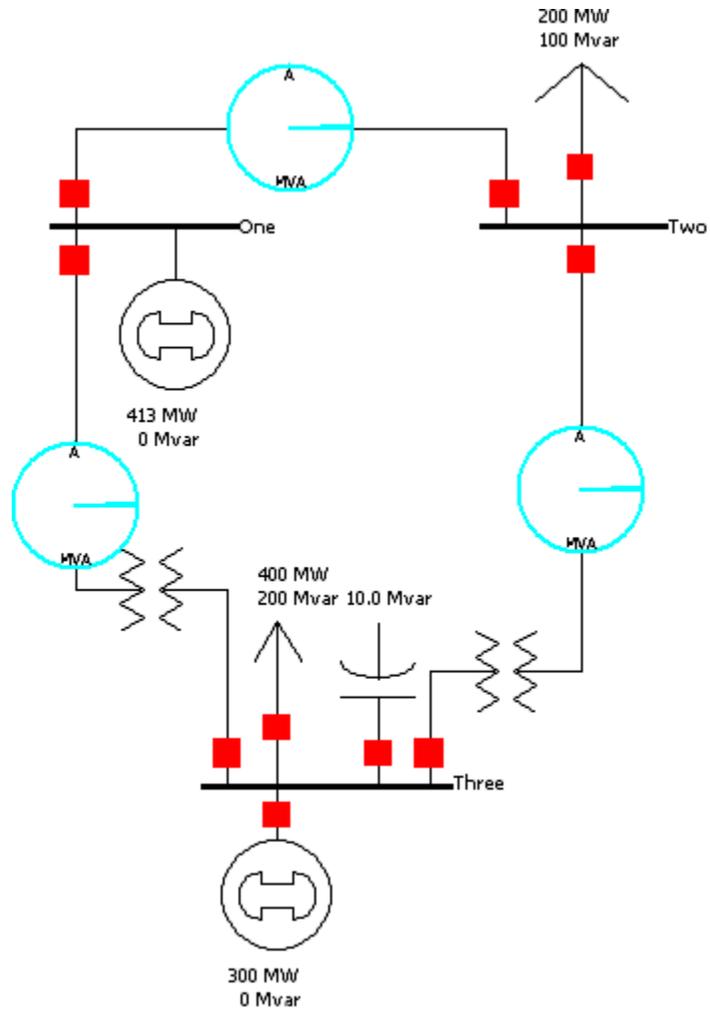
Control Regulation Settings: Voltage Generator Mvar

High Value: 1.0000
Low Value: 0.9900
Reg. Bus #: Target Value: 1.0000

Switched Shunts Blocks

Number of Steps	Mvars per Step

Your oneline should now look similar to the image below.



Tutorial: Inserting Text, Bus and Line Fields Page 11 of 13

Informational fields can be entered directly on the oneline to allow for ease of monitoring when a case is animating.

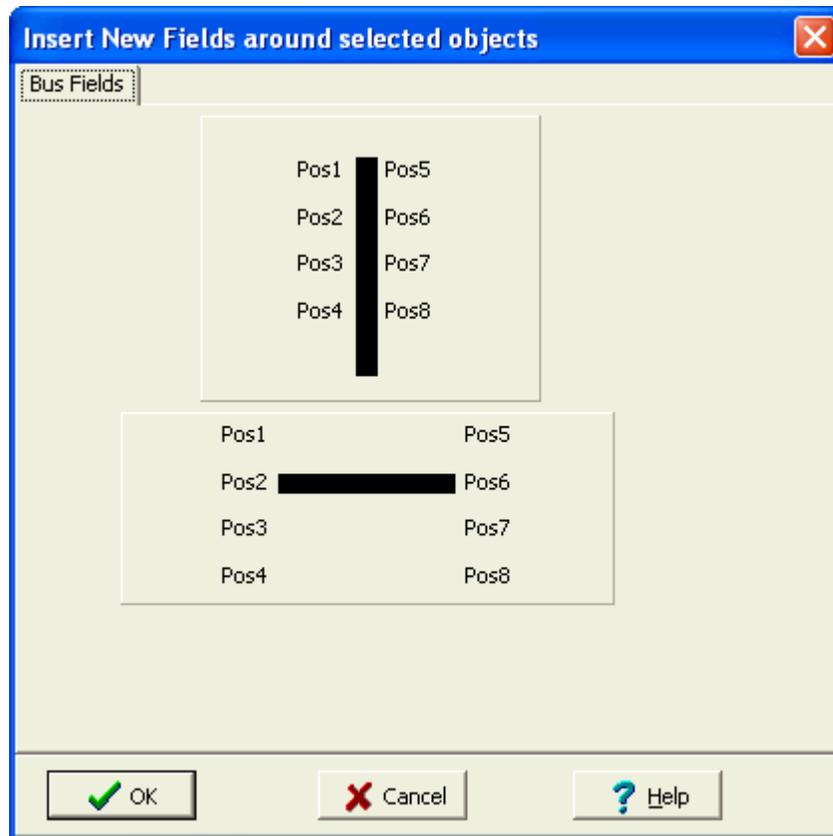
To insert a general text field:

- Select **Background > Text** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Left-click on the oneline in the desired text location to bring up the *Text Object Dialog*. (For this example, left-click in the top center of the oneline.)
- Type the string "First Case" in the *Enter the text* field. Click **OK**.
- To format the text, ensure the text is selected on the one-line then select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This displays the [Font Tab](#) of the [Format Selection Dialog](#).
- Set the font size to **26** and the font color to **blue**.
- To change the text background color, select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab, which summons the Line/Fill Options Tab . If the format menu is already open, click on the [Line/Fill Options Tab](#) of the Format Selection Dialog. Check the *Use Background Fill* box to give the text a white background, and then click *OK*.
- Inserted text can be moved using the same method as any other object on the oneline

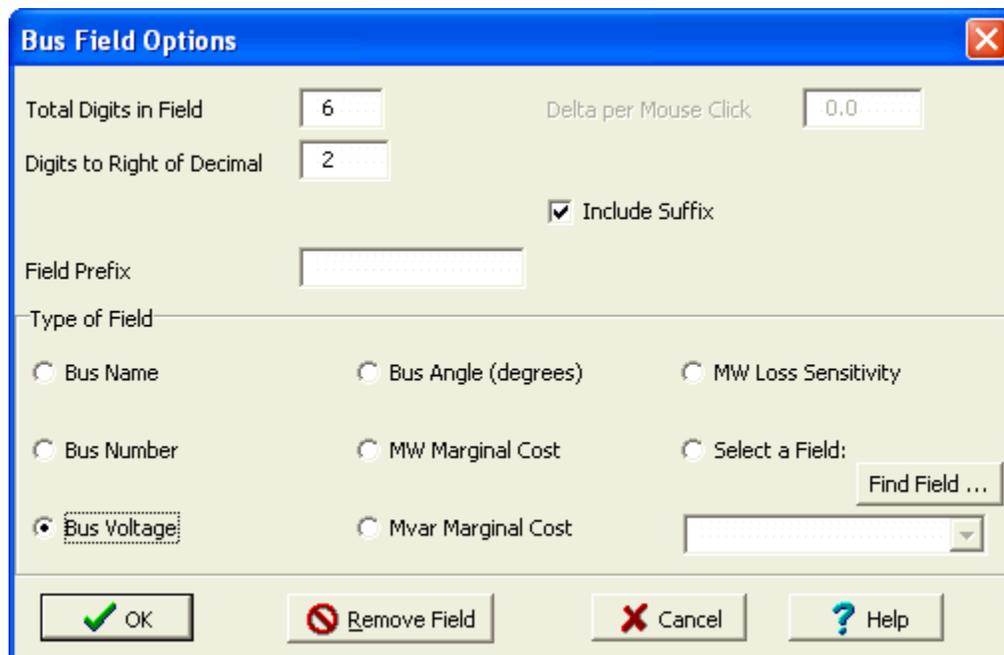
Fields can also display object-related quantities. By default, Simulator has inserted the bus names, generator and load MW and MVAR, and switched shunt MVAR. For this example, we will add a Bus Voltage Magnitude field to each bus and fields showing the power flow on the transmission line and the transformers. Note that object fields can be formatted just like text fields by using the Format menu.

To add additional fields to the display of a particular bus:

- Right-click on the bus to bring up the bus' local menu.
- Select *Add New Fields Around Bus* from the local menu. This opens the [Insert Bus Fields Dialog](#) (pictured below). You may add up to 8 fields per bus. Select the position where you would like to add the new field (position 5 for this example) then click **OK**.



- This opens the Bus Field Options dialog (shown below); select the field **Bus Voltage** to add in the selected position and click **OK**.



- The parameter and position are displayed as highlighted in the Insert New Fields dialog. Click **OK**. Note that the specified bus field has been added to the oneline diagram.
- Repeat this procedure for the other two buses. If necessary, you may move fields manually with the mouse.

Fields can also be inserted using the **Background > Field** option from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Next, we will insert fields showing the power flow at each end of the transmission line.

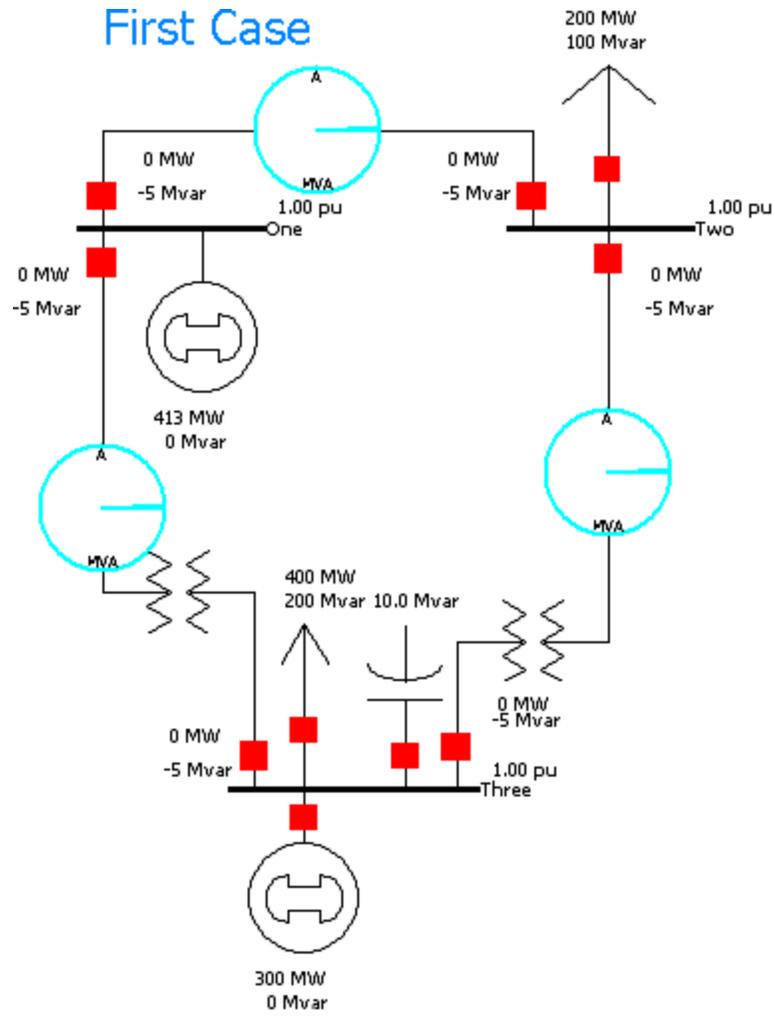
- Select **Field > Transmission Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Line fields show information about transmission lines and transformers. For line fields, flow is always specified at an end of the transmission line or transformer. The end is normally determined automatically by the insertion point.

- Left-click near both bus **1** and the transmission line between buses 1 and 2 in the location you want the power flow text to appear. The *Line Field Options* dialog (shown below) opens automatically.
- The *Near Bus* and *Far Bus* fields should show **1** and **2** respectively. If they do not, enter the correct values.
- Select **MW Flow** then click **OK**. The field is displayed on the oneline in the location you specified. Note that the field can be moved and formatted as previously discussed.
- Select **Field > Transmission Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click near both bus **2** and the transmission line between buses 1 and 2 in the location you want the power flow text to appear. The *Line Field Options* dialog (shown below) opens automatically.
- Now the *Near Bus* and *Far Bus* fields should show **2** and **1** respectively. If they do not, enter the correct values.
- Select **MW Flow** then click **OK**.
- Repeat the procedure to insert **Mvar Flow** fields for the two locations.

We also desire to monitor the MW and Mvar flows on the lines joining buses 1 and 3 and buses 2 and 3 via the transformers. The same commands are used as those used to insert fields for the transmission line.

- Repeat the above steps to insert **MW** (and **Mvar**) **Flow** fields on the lines joining buses 1 and 3 and buses 2 and 3 via the transformers.
- At this point, your first oneline diagram should resemble the one shown below.

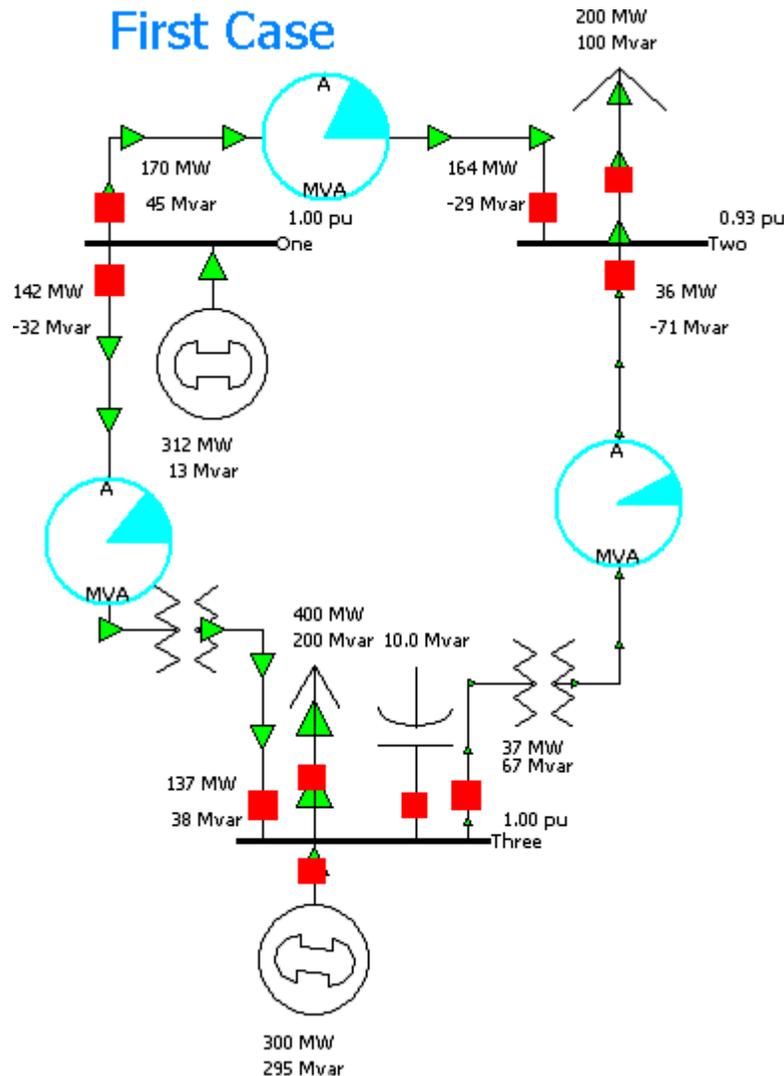


- Save the case.

Tutorial: Solving the Case Page 12 of 13

To solve a case, you must be in run mode:

- Click on **Run Mode** button in the **Mode** ribbon group. Note that if the case has [validation errors](#), a warning will appear. You will need to rectify the problems before you can enter Run Mode.
- Press the **Play**  button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to begin the simulation. Alternatively, to perform a single Power Flow Solution, click the **Single Solution - Full Newton** button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Your case should look similar to the case shown below. If it does, congratulations! You have completed building your first case.



Try clicking on the load circuit breaker to toggle the load's status. A solid red circuit breaker indicates that it is closed, a hollow green box indicates it is open. While the simulation is running, click on the circuit breakers and note the nearly instantaneous change in system flows. If the Log window is

visible, you will get a "backstage" view of what Simulator is doing. Feel free to close the log. To re-open the log, click the **Log** button in the **Log** ribbon group on the [Tools](#) ribbon tab.

With the load circuit breaker closed, open the circuit breaker between bus 3 and its connected generator. Now open any of the transmission line or transformer circuit breakers.

Congratulations, you've just blacked-out your case!

Tutorial: Adding a New Area Page 13 of 13

Next, we will create a second operating area for the case. Large interconnected systems usually have a number of control areas, with each control area responsible for the operation of a particular part of the system. Often, a single control area corresponds to a single owner (such as an investor-owned utility), but it is not unusual for a single control area to have more than one owner. Control areas are connected to neighboring areas through tie lines. A tie line is a transmission line that has one end in one control area and the other end in another. The total amount of power flowing out of a control area is the algebraic sum of the power flowing out on all the area's tie lines. Each control area is responsible for procuring enough power to meet its own load plus losses. The control area can get this power either by generating it itself, or by buying it from another area. This ability to buy and sell power (i.e., power transactions) is one of the principal advantages of interconnected operation.

To create another control area:

- Change to **Edit Mode**. Right click on bus 3 and select **Bus Information Dialog** from the drop down menu.
- Entering a number for an area that does not already exist automatically creates a new area. Enter **'2'** in the *Area Number* field. Enter **'TWO'** in the *Area Name* field. Click **OK**.
- To verify that the case now has two areas, select **Case Summary** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The [Case Information Displays](#) allow you to view the entire case using non-graphical displays. The [Case Summary](#) dialog (shown below) shows the number of buses, generators, lines/transformers, and control areas in the case. You cannot modify any of these values.

CaseSummary for Current Case

Number of Devices in Case			
Buses	3	Series Capacitors	0
Generators	2	2 Term. DC Lines	0
Loads	2	N-Term. DC Lines	0
Switched Shunts	1	Areas	2
Trans. Lines (AC)	3	Zones	1
LTCs (Control Volt)	0	Islands	1
Phase Shifters	0	Interfaces	0
Mvar Controlling	0	Injection Groups	0

Case pathname: C:\Program Files\PowerWorld\Simulator\Cases\firstcase.PW

Case Totals (for in-service devices only)		
	MW	Mvar
Load	600.0	300.0
Generation	611.7	308.3
Shunts	0.0	-10.0
Losses	11.8	18.3

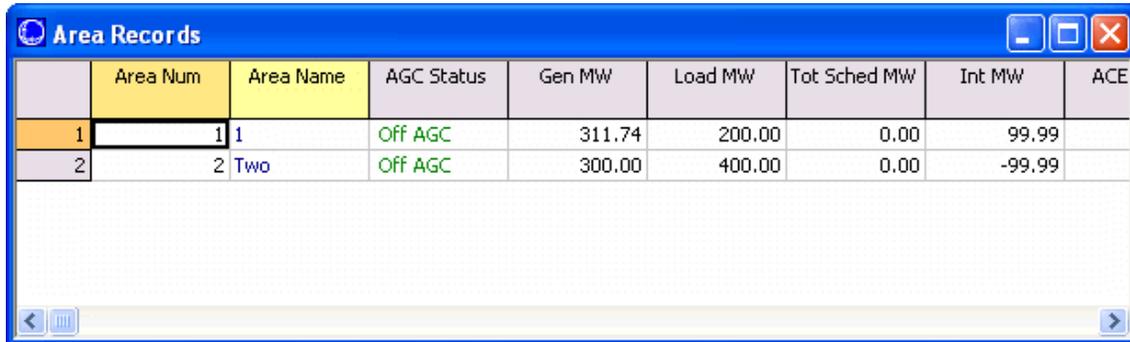
Generator Spinning Reserves		
	Positive [MW]	Negative [MW]
	1388.3	611.7

Slack Buses:
One (1); in Area 1 (1)

Buttons:

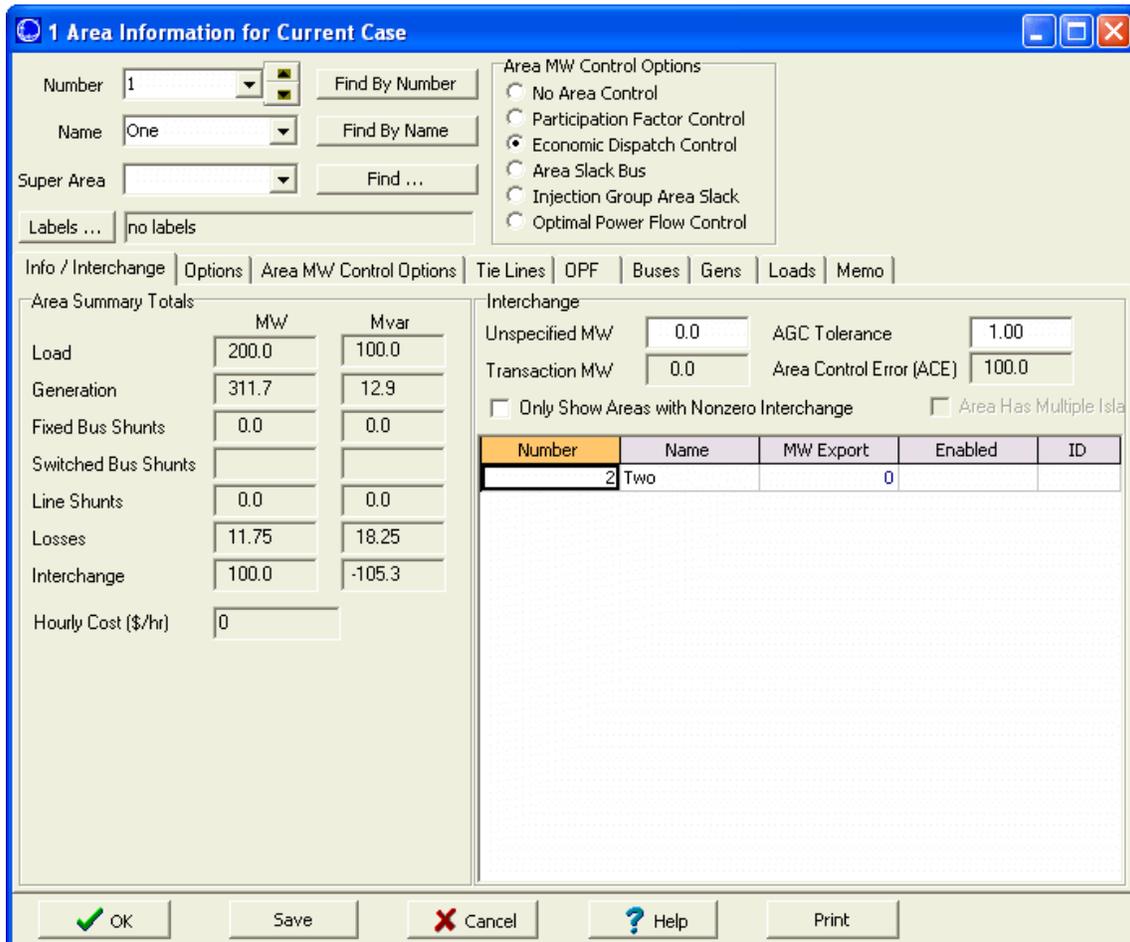
Now we will make sure that both of the control areas are initially set as being on automatic generation control (AGC). AGC insures that the generation in the area is equal to the load plus losses plus and scheduled transactions.

- Select **Case Information > Aggregation > Areas**. The Area display (pictured below) provides a convenient summary of all the control areas in the case. Similar displays exist for buses, generators, lines/transformers, etc.



	Area Num	Area Name	AGC Status	Gen MW	Load MW	Tot Sched MW	Int MW	ACE
1	1	1	Off AGC	311.74	200.00	0.00	99.99	
2	2	Two	Off AGC	300.00	400.00	0.00	-99.99	

- Right-click anywhere on the record for the first area and select the *Show Dialog* option. The [Area Display Dialog](#) is shown (as below).



Number: 1 Find By Number

Name: One Find By Name

Super Area: Find ...

Labels ...: no labels

Area MW Control Options:

- No Area Control
- Participation Factor Control
- Economic Dispatch Control
- Area Slack Bus
- Injection Group Area Slack
- Optimal Power Flow Control

Info / Interchange | Options | Area MW Control Options | Tie Lines | OPF | Buses | Gens | Loads | Memo

Area Summary Totals:

	MW	Mvar
Load	200.0	100.0
Generation	311.7	12.9
Fixed Bus Shunts	0.0	0.0
Switched Bus Shunts		
Line Shunts	0.0	0.0
Losses	11.75	18.25
Interchange	100.0	-105.3
Hourly Cost (\$/hr)	0	

Interchange:

Unspecified MW: 0.0 AGC Tolerance: 1.00

Transaction MW: 0.0 Area Control Error (ACE): 100.0

Only Show Areas with Nonzero Interchange Area Has Multiple Islands

Number	Name	MW Export	Enabled	ID
2	Two	0		

OK Save Cancel Help Print

- Change the Area *Name* to 'One' and set the AGC Status under *Area MW Control Options* to **Economic Dispatch Control**. Select **SAVE** to save this information.

- Next click on the up arrow next to the right of the Area Number field. This displays the Area Record dialog for the next area. Set the AGC Status to **Economic Dispatch Control**, and select **OK** to save your changes and close the dialog.
- Note the *AGC Status* field in the **Area Records Display** now shows **ED** in the *AGC Status* fields. Close the Area Records display.

Starting with an Existing Case Page 1 of 15

This procedure describes how to create a power system model from an existing power flow file using PowerWorld Simulator. This tutorial assumes that you have at least some familiarity with PowerWorld Simulator. If you need a more general introduction, please see [Creating a New Case tutorial](#).

PowerWorld cases can be easily created from existing power flow cases stored in PTI RAW versions 23-30, GE PSLF text format (EPC version 11.X and later), and IEEE common format.

Simulator provides a static model of a power system. The power flow data is a subset of the system model. For some studies this model is sufficient. For other studies the model needs to be augmented by adjusting: generator cost information, the reactive capability curve, PowerWorld Simulator case options, interface definitions, injection group definitions, contingency definitions, the time variation of the load, etc....

To begin, double-click on the PowerWorld Simulator icon. This starts Simulator. In this example we will be building a case from an existing power flow file.

Tutorial: Loading an Existing Power Flow File Page 2 of 15

Once Simulator has been started, set Simulator to Edit Mode by clicking on the **Edit Mode** button. Select **Open Case** from the [Application Button](#). An **Open** dialog box will appear on the screen. To select a power flow file, click on the **Type of File** field in the lower left hand corner of the dialog box. A list of file formats will appear. Choose the appropriate format, and the available files of the type selected will appear in the box above **Type of File**. Choose the desired file from the list of available files and select *OK*. **In this example, we will be building a oneline for the 3990 bus MAIN 1998 summer case saved in PTI version 23 format as mdb98s.raw. When prompted to create a oneline, select 'yes'.** Your display should immediately turn white. This shows the blank background upon which you will be drawing the oneline diagram for your case. Onelines are used in power system analysis to represent the actual three-phase power system using a single line to represent each three phase device. The information from the power flow file is now accessible to Simulator.

Tutorial: Case Information Displays Page 3 of 15

Simulator presents many different [Case Information Displays](#) to provide a text-based view of the case. For example, to view or modify the description of the case, select **Case Description** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The **Case Description Dialog** should appear, showing the text description of the power flow case. In PowerWorld, this text description may be arbitrarily long.

You can also obtain a case summary by selecting **Case Summary** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The summary dialog will appear. It provides a summary of the case and of the total case load and generation. None of the fields in this dialog can be changed, as they are intended for informational purposes only.

The Area/Zone Filters information display is another important article. The [Area/Zone/Owner Filters](#) feature restricts the contents of other case information displays to certain areas or zones. This is particularly useful for large cases. Open the Area/Zone Filters display by selecting **Area/Zone Filters** from the **Case Information** ribbon group on the [Case Information](#) ribbon tab. As with all Case Information Displays, you can click on a column heading to sort the list on a particular field; click on that same column heading again to reverse the sort order. To change a particular area's area/zone filter status, simply left-click on the area's area/zone filter status field. To change the filter status for all areas in the case, right-click on the display to bring up its local menu (all case information displays have a local menu), and select either **Toggle All Yes** or **Toggle All No**. *For our example, set all areas to 'no' except WUMS.*

Another important text-based display is the Power Flow List. The [power flow list](#) shows the complete power flow information for all areas whose area/zone filter is set to 'yes'. To display the power flow list, select **Power Flow List** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The power flow list has other options that can be accessed by right-clicking anywhere on the list. If you wish to view more details about a particular device, you can do so by holding down the **Ctrl** key and left-clicking on the device. To move through the list of buses, you can use the arrow keys or the scroll bar. If you are viewing a particular bus and you wish to view a bus connected to it, double-click on the line connecting the two buses to view the flows at the other bus. Note that some of the transmission lines have a circuit identifier of '99'. Whenever a '99' is used for an identifier, it usually means that the transmission line is an equivalent line.

A shorter version of the power flow list can often be more useful. Such a display is offered by the Quick Power Flow List. The [Quick Power Flow List](#) is similar to the power flow list except that it shows flows at individual buses or a set of buses, regardless of area/zone filter settings. You can view flows at any bus in the case. To display the quick power flow list, select **Quick Power Flow List** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. Identify the buses of interest in the **Bus Number** field by entering the number of either a single bus, a set of buses separated by commas, or a range of buses specified using a hyphen. Again you can double-click to move to a desired bus, or choose **Show Object Dialog** from the display's local menu to see the information dialog for any object.

Tutorial: Solving the Case Page 4 of 15

To solve the power flow case we have been using in this example, press the **Single Solution - Full Newton** button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab, or press the Single Solution icon on the [Quick Access](#) toolbar. You may wish to show the Message Log before you solve the case so as to monitor the solution process. When you choose to perform a Single Solution, the application automatically switches to Run Mode if it is not already there. The system has initial mismatches because of voltage truncation in the power flow file. The case should converge quickly, perhaps in 2 or 3 iterations. After the single solution has been performed, Simulator now has the solved power flow in memory, and you are ready to build the oneline.

It is not necessary that the power flow case be solved before you create the oneline. However, we recommend that you solve the existing case first to make sure that it is valid.

Tutorial: Building Onelines Page 5 of 15

PowerWorld Simulator makes the power system case easier to analyze by presenting results visually using a oneline diagram. You do not need to represent every bus in the power flow model on the oneline; a oneline diagram need be created only for the desired portion of a system under study. Simulator can automatically link the constructed oneline diagram to the existing power system model. The following sections of this tutorial will discuss placing various visual components onto a oneline diagram. In all of the following sections the actions described will relate to the aforementioned MAIN example.

Tutorial: Entering a Bus Page 6 of 15

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices connect together. To build a oneline diagram, you draw the buses, attach devices such as generators and loads to the buses, and connect the buses together with transmission lines and transformers.

To begin entering devices onto the blank oneline, you must first switch to Edit Mode. If you are not already in Edit Mode, switch back to Edit Mode now. Show the Quick Power Flow List and move it towards the bottom of the screen. Show bus 39820 by typing that number in the **Bus Number** field. Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the oneline towards the top center to define the point at which the new bus will be added. The Bus Information Dialog will appear. In the **Bus Number** field enter 39820. Select **Find by Number** to view the bus information. You should see the information appear in the Bus dialog fields that corresponds with the MAIN power flow case. Select *OK* to place the bus. The bus should now appear on your screen.

If it has not already been done for you automatically, add a bus field identifying the new bus' number immediately to the left of the new bus. To do this, click on or to the left of the display object that represents bus 39820 and select **Field > Bus Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. The Bus Field Options Dialog will appear for you to fill out. Designate the type of field as *Bus Number* and close the dialog. Simulator will add a text object showing the bus number at the point where you had clicked.

Repeat this procedure to place buses 39881 and 39821 on the oneline, along with their bus numbers.

Tutorial: Automatic Line Insertion Page 7 of 15

Transmission lines between buses can be inserted manually by choosing **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and clicking on the beginning bus. Vertices may be defined along the way by clicking the mouse on the diagram where vertices should appear. However when creating a case from an existing power flow file, you also have the option to insert transmission line display objects automatically. To do this, select **Auto Insert > Lines** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. Accept the default options and click OK. The lines joining the visible buses on the display are automatically added, along with circuit breakers and pie charts, provided those options are set. Simulator will draw only transmission lines that link buses that have already been drawn on the oneline diagram. If you add another bus to the diagram, you can again auto-insert lines, and Simulator will only insert lines that are not already present on the display.

Panning and Zooming Page 8 of 15

Two features of Simulator are indispensable when you have a large, detailed oneline: panning and zooming. To pan from side to side or up and down, either use the arrow keys or the scrollbars on the sides of the oneline. To zoom in or out of the oneline, hold the **Ctrl** key down and press the up arrow to zoom in and the down arrow to zoom out. For the example, pan up and insert buses 39819 and 39841, again with bus fields showing their numbers. Again use **Auto Insert > Lines** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab to add the transmission lines.

Tutorial: Adding Background Page 9 of 15

Sometimes you may wish to insert background elements on a oneline such as bodies of water or state lines to convey geographic location. To show static background elements on the oneline, select **Background > Background Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the diagram to start the background line, and click at other points on the diagram to add segments. Double click to terminate the background line. If you wish to fill in the area inside the background line or change the background line's color, select the background line and choose **Line/Fill** from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab and make the appropriate choices in the resulting dialog. You may wish to experiment now with drawing background lines or objects and with adding fill color. Additionally, some political and geographic borders can be inserted by selecting **Auto Insert > Borders** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab.

Tutorial: Simulating the Case Page 10 of 15

Once you have constructed a oneline diagram showing the area of interest, you can simulate the case. Simulator does not only show the magnitude and direction of flows on transmission lines, but can also *animate* them. To configure the animated flows, switch to Edit Mode and choose **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab. This dialog box also allows you to change the size, density, and fill color of the animated flows for easier visualization. At this point, just click *OK*. Next, save your case by selecting **Save Case** from the [Application Button](#) or by pressing the

Save Case  button on the [Quick Access Toolbar](#). If you have not already saved the case, a **Save As** dialog will prompt you to select a name. Both the oneline diagram and the case will use this name.

To begin solving and animating the load flow solution, switch to Run Mode and then press the Play button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. You should now see Simulator modeling the 3990-bus case.

Tutorial: Run-time Object Dialogs Page 11 of 15

While the animation ensues, you might want to view or change some study parameters. To do this, first pause the animation. (Note that you do not *have* to pause the animation to tweak parameters. You can adjust anything in the case while the animation runs, too.) Then, right-click on any of the objects on the display, and choose Show Dialog from the popup menu. This will bring up the run-time dialog for the object. Many of the parameters on this dialog can be modified, and the new settings will take effect when the animation is restarted.

Online Local Menu Page 12 of 15

Several options are available at run time from the oneline diagram's local menu. To call up the menu, right-click on an empty portion of the oneline. The local menu will appear. Use it to print the oneline, save it as a metafile, or copy it to the clipboard. You can also find a particular bus on the oneline, access panning and zooming options, set oneline display options, view information about the power system area in which you clicked, create a contour plot, and use the difference flows activity, all from the oneline local menu.

Tutorial: Area Page 13 of 15

Often, system data is most conveniently displayed by area. To view the Area Records Display, select **Aggregation > Areas** from the [Model Explorer](#). The resulting display summarizes information about all the areas in the case. You can sort the entries by clicking on the column labels.

Limit Violations Page 14 of 15

You can view a report of limit violations by selecting **Limit Monitoring Settings** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab. A display showing bus voltage violations, line/transformer violations or interface violations will appear. To show only violations, select **Violating Elements** from the **Elements to Show** options. If the **Use Area / Zone Filters on List Displays** is checked, then the displayed violations will correspond only to areas whose **Shown** field is set to **Yes** on the [Area/Zone/Owner Filters](#) display.

If you wish to see more information on a bus that appears in the violation list, right-click on the bus number and select [Quick Power Flow List](#) from the resulting local menu. If the [Case Information Display](#) is set to refresh automatically, the list of bus voltage violations will update as new violations occur.

Other Case Information Displays Page 15 of 15

In addition to the few displays discussed in this tutorial, Simulator offers many other [Case Information Displays](#). In most cases, you can view information about buses, generators, lines, transformers, loads, and zones simply by right-clicking on the object in question and choosing the appropriate option from the object's local menu. The best way to become more familiar with the displays and the information contained in them is simply to play with a Simulator case and online. Simulator's interface has been designed to be simple and intuitive. If you run into problems, consult the On-line Help for assistance.

Tutorial: Solving an OPF Page 1 of 6

Note: The OPF option in PowerWorld Simulator is only available if you have purchased the OPF add-on to the base package. Please contact PowerWorld Corporation at info@powerworld.com or visit the website at <http://www.powerworld.com> for details about ordering the OPF version of Simulator.

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the standard mode, Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. With the optimal power flow (OPF) enhancement, Simulator OPF can also solve many of the system control equations using an Optimal Power Flow algorithm. Specifically, Simulator OPF uses a linear programming (LP) OPF implementation.

The purpose of an OPF is to minimize an [objective \(or cost\) function](#) by changing different [system controls](#), taking into account both [equality and inequality constraints](#). These constraints are used to model the power balance constraints and various operating limits.

In Simulator OPF, the LP OPF determines the optimal solution by iterating between solving a standard power and then solving a linear program to change the system controls to remove any limit violations. See [OPF Primal LP](#) for more details.

Tutorial: OPF Three Bus Example Page 2 of 6

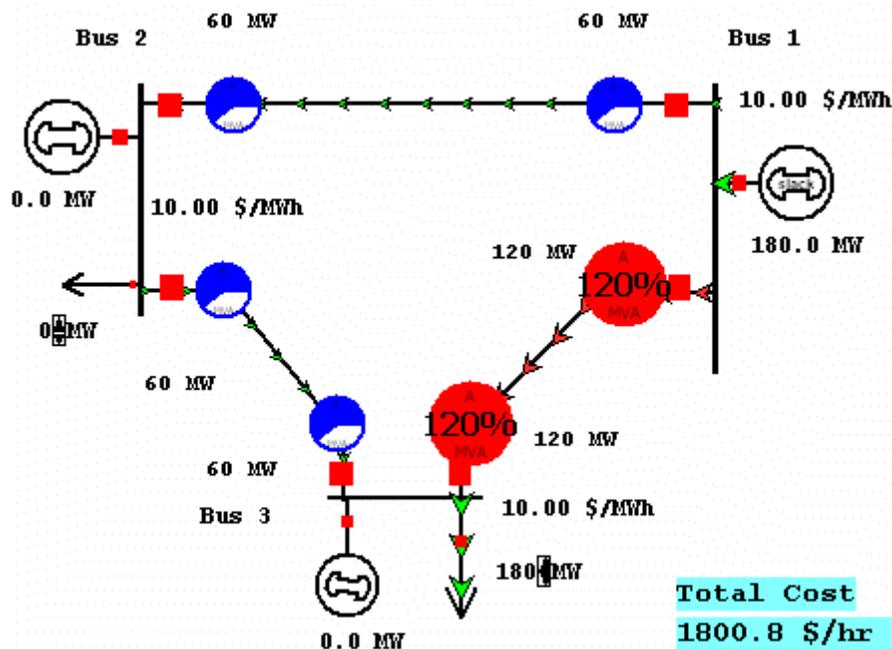
For this tutorial, we will start with a provided three-bus case (B3LP) found in the *PowerWorld\Simulator\Sample Cases* directory. Bus 1 is the system slack bus. All buses are connected via 0.1 pu reactance lines, each with a 100 MVA limit. There is a single 180 MW load at bus 3. The generator marginal costs are:

- Bus 1: 10 \$/MWhr; Range = 0 to 400 MW
- Bus 2: 12 \$/MWhr; Range = 0 to 400 MW
- Bus 3: 20 \$/MWhr; Range = 0 to 400 MW

To begin:

- Load B3LP case.
- Verify the above system specifications by right clicking on each line, bus, and generator and selecting its respective **Line/Bus/Generator Information Dialog** from the drop down menu.
- Select **Run Mode**.
- To be included in the OPF, all required area AGC status fields must be selected to OPF. To set area AGC to OPF: Select **OPF Case Info > OPF Areas** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The OPF Area Records display automatically opens. Verify the AGC Status field is set to **OPF**. If it is not, double click on the field to change its value. Close the OPF Area Records display.
- Select **Add Ons > Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab to solve the case. Note: line limits are not initially enforced.

Your display should look similar to the following:



B3LP Solved Using Add Ons > Primal LP

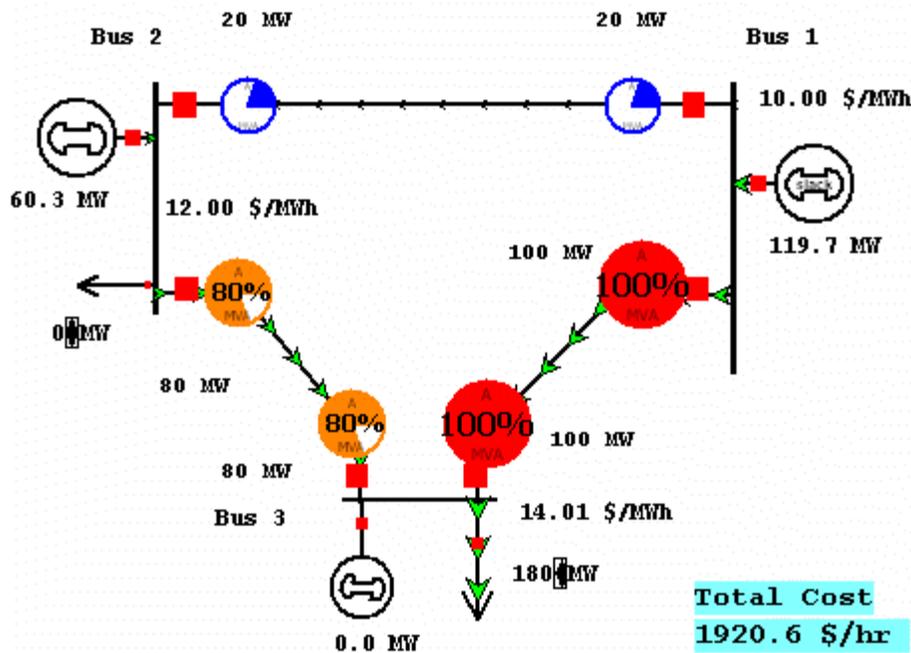
Note the line from Bus 1 to Bus 3 is overloaded and that all buses have the same marginal cost.

Tutorial: OPF Line Limit Enforcement Page 3 of 6

- Select **OPF Case Info > OPF Areas** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.

The *Branch MVA* column specifies whether or not the MVA limits should be enforced for transmission lines and transformers that have at least one terminal in this area. For a transmission line or transformer to be included in the OPF constraints, *Line/Transformer constraints* must not be disabled on the **OPF Options and Results Dialog**, and the individual line/transformer must be enabled for enforcement on the **OPF Line/Transformer MVA Constraints** display.

- Double click on the *Branch MVA* field to change the value to **YES**. Close the OPF Area display
- Select **OPF Options and Results** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.
- The **LP OPF Dialog** opens automatically. Select the **Constraint Options** tab then click *Disable Line/Transformer MVA Limit Enforcement* to remove the checkmark.
- Click **Solve LP OPF** then click **OK**.



B3LP Solved with Line Limit Enforcement Enabled

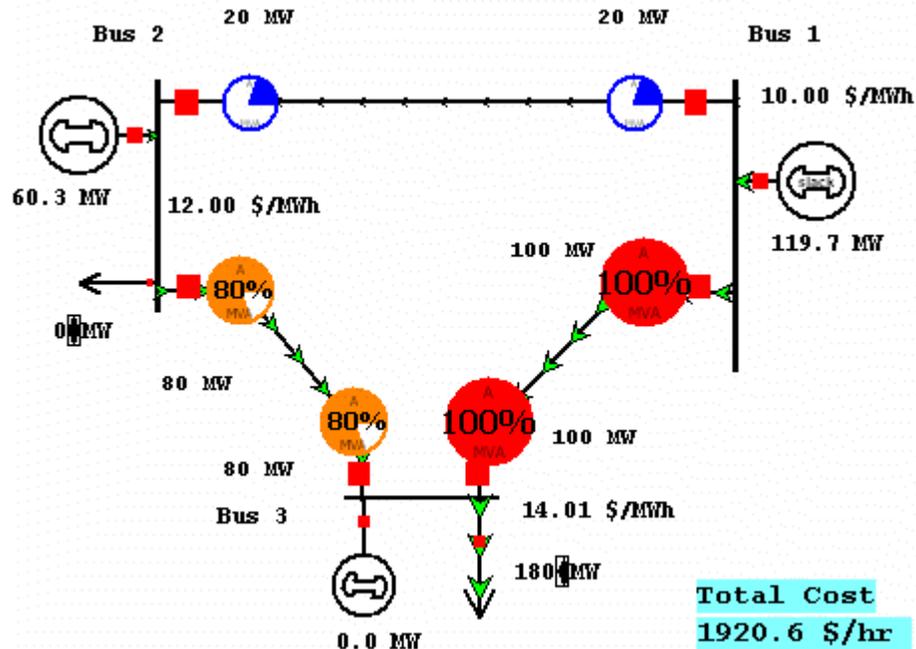
The LP OPF re-dispatches to remove the line limit violation. Bus marginal costs have changed from the initial example. The new bus marginal cost at Bus 3 is 14 \$/MWh. To verify this:

- Increase the load at Bus 3 by 1 MW. Right click on the load, enter "181.0" in the *Constant Power / MW Value* field in the **Load Options** dialog. Click **OK**.
- Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.

The Total Cost increased to 1935 \$/hr from the previous value of 1921 \$/hr, a difference of 14 \$/hr when Bus 3 load was increased by 1 MW.

Tutorial: OPF LMP Explanation Page 4 of 6

Explanation of Bus 3 LMP = 14 \$/MWh (from previous page):



B3LP Solved with Line Limit Enforcement Enabled

All lines have equal impedance. Power flow in a simple network distributes inversely to impedance of the path.

- For Bus 1 to supply 1 MW to Bus 3, 2/3 MW will take the direct path from 1 to 3, while 1/3 MW will take the path from 1 to 2 to 3.
- Likewise, for bus 2 to supply 1 MW to Bus 3, 2/3 will go from 2 to 3, while 1/3 will go from 2 to 1 to 3.
- To supply one additional MW to Bus 3, we need the change in power of generator 1 (Pg1) plus the change in power of generator 2 to equal 1 MW.

$$Pg1 + Pg2 = 1 \text{ MW}$$

- With the line from 1 to 3 limited, no additional power flows are allowed on it.

$$(2/3)Pg1 + (1/3)Pg2 = 0$$

- Solving the above system of equations results in:

$$Pg1 = -1 \text{ MW and } Pg2 = 2 \text{ MW}$$

$$\Delta\text{cost} = \sum(\Delta Pg * LMP) = [(-1 \text{ MW}) * (10.00 \text{ $/MWh}) + (2 \text{ MW}) * (12.00 \text{ $/MWh})] = \mathbf{14.00 \text{ $/hr}}$$

Tutorial: OPF Marginal Cost of Enforcing Constraints Page 5 of 6

Similar to the bus marginal cost, you can also calculate the marginal cost of enforcing a line constraint. For a transmission line, this represents the amount of system savings that could be achieved if the MVA rating was increased by 1.0 MVA.

- Select **OPF Case Info > OPF Lines and Transformers** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The OPF Constraints Records dialog opens.

Note the column displaying **MVA Marginal Cost** displays **6.0** for the line from Bus 1 to Bus 3. This is determined based on the following:

- With no change in system load:

$$Pg1 + Pg2 = 0$$

- If we allow one additional MVA to flow on the line from Bus 1 to Bus 3:

$$(2/3)Pg1 + (1/3)Pg2 = 1$$

- Solving the above system of equations results in:

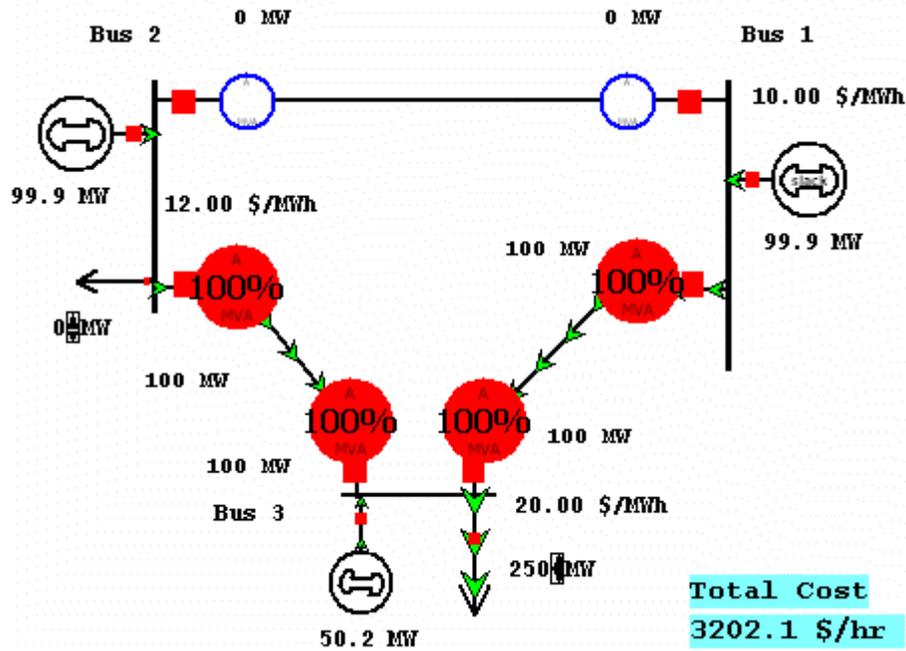
$$Pg1 = 3 \text{ MW and } Pg2 = -3 \text{ MW}$$

$\Delta\text{cost} = \Sigma(\Delta Pg * LMP) = [(3 \text{ MW}) * (10.00 \text{ \$/MWh}) + (-3 \text{ MW}) * (12.00 \text{ \$/MWh})] = \mathbf{-6.00 \text{ \$/hr}}$, a net savings of 6.00 \$/hr.

Tutorial: OPF Unenforceable Constraints Page 6 of 6

Next we will consider a case with unenforceable constraints. To begin, increase the load at Bus 3 to 250 MW then run the LP OPF again:

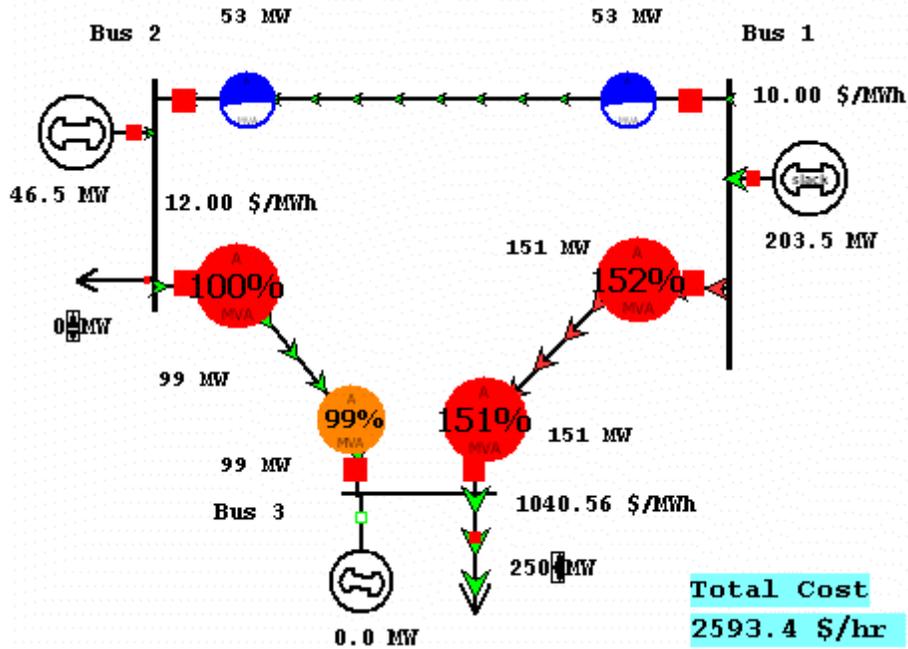
- Right-click on the load at Bus 3. Enter 250 in the Constant Power / MW Value field.
- Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.



B3LP with Bus 3 Generator Carrying Load

Note that the transmission lines connecting Bus 3 to the other buses are both at their respective MVA limit and that the generator at Bus 3 is supplying the load in excess of the transmission line limits. Next we will open the generator at Bus 3 thereby inserting an unenforceable constraint:

- Left-click on the red breaker symbol connecting the generator to Bus 3 to open the breaker.
- Select **Primal LP LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.



B3LP Solved with Unenforceable Constraints

Both constraints cannot be enforced. If a constraint cannot be enforced due to insufficient controls, the slack variable associated with enforcing that constraint *can not* be removed from the LP basis.

Note the new LMP value for bus 3 exceeds 1000 \$/MWh. The marginal cost depends on the arbitrary cost of the slack variable. This value is specified in the *Marginal Violation Cost* field of the **Add Ons > OPF Options and Results** dialog. See [OPF Options and Results dialog](#) for more details.

Additional Example

To see another OPF example select **PowerWorld Simulator Add-on Tools > Optimal Power Flow (OPF) > Examples** from the table of contents in the on-line help file.

Tutorial: Contingency Analysis

This tutorial will walk you through the basic commands necessary to insert contingencies and have Simulator automatically analyze the results. Please see [Introduction to Contingency Analysis](#) for the necessary background information regarding the capabilities and uses of the Contingency Analysis tool.

For this tutorial, we will use an existing 7-bus case.

- Open case B7SCOPF from the *Program Files/PowerWorld/Simulator/Sample Cases* directory.
- Ensure Simulator is in [Run Mode](#).
- Select **Contingency Analysis** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. Simulator opens the Contingency Analysis Dialog.

The next section of the tutorial discusses Defining Contingencies and provides an example of inserting a single element contingency.

1 2 3 4 5 6 7 8 9 10 11

Next

Index

A

- Abort, 27, 53
- Absolute Value, 71
- ACE, 598, 599, 701, 709, 739
- Actions, 1284, 1299, 1304, 1312, 1316, 1318, 1320, 1321, 1323
- Add-On Tools, 927, 986, 1039, 1121
- Admittance, 152, 663
- Advanced, 543, 544, 546
- Advanced Filter, 531, 542, 543, 544, 546, 547, 557
- Advanced Sort, 522, 527
- AGC, 597, 599, 610, 667, 668, 676, 701, 737, 739
- Aggregation, 39
- Aggregations, 504
- Align Group Objects, 322
- Anchored, 37, 73, 159, 311, 321
- Anchored Objects, 159, 245, 321
- Anchors, 159, 245
- Angle Smoothing, 698
- Animated Flows, 74, 83
- Animation, 66, 74, 83, 857
- Appending, 17, 244
 - Case, 17
- Appending a Case, 244
- Apply Template, 68
- Area, 3, 50, 169, 172, 205, 302, 308, 349, 436, 460, 466, 467, 468, 469, 470, 471, 541, 597, 599, 603, 710, 712, 713
 - Zoom, 50, 308
- Area Average Cost, 713
- Area Chart, 710
- Area Control, 701
- Area Control Error, 701, 709
- Area Display, 597, 599
- Area Display Objects, 161
- Area Display Options Dialog, 162
- Area Fields, 169, 172, 316, 471
- Area Information, 460, 462, 463, 464, 465, 466, 467, 468, 469, 470
 - Area Buses, 462
 - Area Gens, 463
 - Area Loads, 464
 - Area MW Control Options, 465
- Area Loss Sensitivity, 616
- Area Losses, 711
- Area MW Control Options, 465
- Area MW Transactions, 712
- Area Slack, 702
- Area Slack Bus Control, 702
- Area Transactions, 597, 599, 712
- Area/zone, 308
- Area/Zone/Owner, 541
- Area/Zone/Owner Filters, 30, 527, 531, 541
- Areas, 456, 460, 466, 467, 468, 541, 591, 597, 599, 634, 701, 709, 710, 711, 712, 713
- Associate, 1412
- ATC, 19, 861, 1077, 1078, 1081, 1088, 1093, 1098, 1106, 1108, 1318, 1334
 - Multiple Scenario, 1078, 1098
- ATC Analysis, 1098
- ATC Analysis Methods - Iterated Linear Step IL, 1114
- ATC Analysis Methods - Iterated Linear Step IL then Full CTG Solution, 1117
- ATC Analysis Methods - Single Linear Step SL, 1113
- ATC Analysis Methods - Solution Methods, 1112
- ATC Dialog Analysis Tab, 1096
- ATC Dialog Result Tab, 1095
- ATC Dialog Options Tab, 1080
- ATC Extra Monitors, 1093
- Augmenting Max Flow Min Cut, 296, 297
- Auto Insert, 203, 274, 275, 277, 278, 279, 280, 281, 282, 751, 866, 868

- Borders, 282
 - Contingencies, 751
 - Display Objects, 203, 275, 277, 278, 279, 280, 281
 - PTDF Directions, 866, 868
 - Auto Insert Contingencies, 820
 - Auto Insert Injection Groups, 488
 - Auto Size, 526
 - Column Widths, 526
 - Auto Updating, 308
 - Auto Zoom, 308
 - Automatic Control, 379, 405, 443
 - Automatic Generation Control, 737
 - Automatic Line Tap Dialog, 257
 - Automatically Insert, 203, 274, 275, 277, 278, 279, 280, 281, 751, 868
 - Borders, 282
 - Contingencies, 751
 - Display Objects, 203, 275, 277, 278, 279, 280, 281
 - PTDF Directions, 866, 868
 - Automatically Open, 692
 - Autosize, 526
 - Auxiliary File
 - Export Format Description, 568
 - Auxiliary Files, 141, 142, 143, 146, 147, 148, 754, 1183, 1278
 - Formats, 142, 143, 146, 147, 148, 754
 - Available Contingency Actions, 741
 - Available Transfer Capability, 19, 861, 1077, 1078, 1098, 1108, 1334
 - Average Cost, 713
 - AVR, 379, 405, 443, 606, 610
- B**
- Back, 38, 312, 317
 - Send, 38, 312, 317
 - Background, 38, 39, 71, 213, 215, 216, 217, 307, 308, 312, 317, 318
 - Color, 71, 307
 - Pictures, 217
 - Shapes, 213, 215, 216
 - Background Objects, 213, 215, 216, 217, 221, 222, 224, 316
 - Background_Lines_Dialog, 214
 - Base, 38, 312, 317, 403, 1110
 - Level, 317
 - Base Case, 730, 731, 732, 733, 735, 1110
 - Base Case Violations, 781
 - Binding Constraint Dialog, 898
 - Bitmap format, 68
 - Blink, 72
 - Blocks, 759
 - BMP, 69
 - Branch, 71, 308, 634, 1108
 - Object, 308
 - Branch Limiters, 1108
 - Branches, 596, 654
 - Branches that Create Islands, 721
 - Breakpoint, 484
 - Bring, 302, 317, 540
 - Front, 38, 312, 317
 - Browse, 268
 - Browsing, 72
 - Build, 136, 137
 - Building New Case, 136
 - Bus, 3, 35, 37, 67, 71, 82, 159, 174, 175, 176, 246, 247, 249, 251, 252, 253, 262, 300, 308, 311, 323, 328, 331, 333, 423, 605, 606, 608, 714, 717, 789, 835
 - Object, 308
 - Bus Field Information, 331
 - Bus Fields, 175, 316, 331
 - Bus Information, 397, 423
 - Bus Information Dialog, 526
 - Bus Load Throw Over Records, 789
 - Bus Mismatches, 608
 - Bus Options, 328
 - Bus Renumbering, 246, 247, 249
 - Bus View, 537, 714, 717
 - Buses, 50, 159, 174, 246, 251, 252, 253, 255, 274, 328, 333, 397, 423, 456, 591, 596, 658, 714, 717, 773, 835

Buyer, 1078, 1093

C

Calculate Participation Factors, 739

Calculated Fields, 515

Cascade, 63, 117

Case, 17, 19, 67, 129, 132, 133, 135, 138, 244, 526, 590, 591

 Appending, 17, 244

 Submode, 19

Case Description, 590

Case Formats, 129, 590

Case Information, 17, 20, 507, 508, 518, 522, 523, 526, 581, 590, 593, 595, 596, 597, 599, 608, 619, 644, 689

Case Information Customizations, 508, 517

Case Information Displays, 20, 318, 507, 508, 517, 518, 522, 523, 527, 541, 542, 543, 544, 546, 547, 548, 549, 550, 553, 555, 556, 560, 564, 565, 567, 581, 590, 597, 599, 605, 609, 619, 689

 Custom Field Descriptions, 512, 513

Case Information Displays Colors and Cell Styles, 520

Case Information Displays Using Cell Handles, 521

Case Information Filterbar, 505, 531, 543, 544, 546, 547

Case Information Toolbar, 505, 508, 526

Case Pathname, 592

Case References, 745

Case Summary, 591

Case Templates, 68

Case Totals, 592

Case Validation, 132

Center, 50, 111, 308

 Display, 50

 online, 308

Change, 307

Change button, 316

Change Field Data, 583

Change Fill Color, 316

Change Line Color button, 316

Change Timepoint Time Dialog, 907

ChangeParametersMultipleElement Function, 1133

ChangeParametersMultipleElement Sample Code, 1134

ChangeParametersMultipleElementFlatInput Function, 1136

ChangeParametersMultipleElementFlatInput Sample Code, 1137

ChangeParametersSingleElement Function, 1130

Charts, 709, 710, 711, 712, 713

Choosing, 317

 Layer, 317

Circuit, 622

Circuit Breakers, 186

Close Online, 140

Closest Facilities to Point, 110

Color, 37, 71, 75, 305, 307, 311, 316, 850

Color Dialog, 307, 316

 displays, 316

Color Key, 565

Color Map, 850

Colors, 313, 315, 316, 565

Column Metrics, 540

Column Widths, 526

 Auto Size, 526

Comments, 21

Complete Show, 71

Conditional Display, 66, 119, 319

Conditional Display by Zoom Level, 319

Configuring, 527

 Case Information Displays, 527

Connect, 1124

Container Window, 13, 20, 62, 116

Context-sensitive, 11

Contingencies, 750

Contingency, 19, 743, 744, 745, 751, 754, 755, 757, 759, 760, 762, 763, 764, 766, 769, 771, 773, 775, 777, 778, 780, 783, 787, 789, 790,

- 792, 793, 794, 796, 799, 802, 813, 814, 815, 816, 1469
 - Blocks, 783
 - Case References, 745
 - Compare, 764
 - Delete, 764
 - Global Actions, 783
 - Records, 745, 757
 - Report, 764
 - Submode, 19, 1316
- Contingency Action, 743
- Contingency Actions, 764, 771, 775, 799
 - Compare Contingencies, 764
- Contingency Analysis, 3, 636, 740, 743, 744, 745, 750, 751, 754, 757, 759, 760, 762, 763, 764, 766, 769, 771, 773, 775, 777, 778, 781, 784, 786, 789, 790, 792, 793, 794, 796, 799, 802, 813, 814, 815, 816, 861, 878, 1078, 1469
 - Advanced Limit Monitoring, 780
 - Advanced Modeling, 783
 - Auto Insert Contingencies, 751
 - Compare Contingencies, 764
 - Load Contingencies, 754
 - Options, 777, 778
 - Reference State, 745
 - Saving Records, 757
 - Summary, 793
- Contingency Analysis Defining Contingencies, 750
- Contingency Analysis Definitions, 743
- Contingency Analysis Results, 764, 794
- Contingency Analysis Tutorial, 815, 816, 1469
- Contingency Blocks, 759, 760, 783
- Contingency Case References, 745, 748, 749
- Contingency Case References - State Information, 746
- Contingency Definition, 760, 766, 771, 773, 775
- Contingency Element, 743
- Contingency Element Dialog, 760, 771, 775, 802, 813
- Contingency Global Actions, 783
- Contingency Records Auto Insert, 819
- Contingency Specific Options, 744
- Contingency Violations, 769, 773, 796
- Contingency Violations Display, 767
- Continuously Update, 565
- Contour, 68, 564, 565, 842, 843, 844, 848, 850
- Contour Column, 540, 564, 565
- Contour Type, 565, 844, 848
- Contouring, 518, 564, 565, 842, 843, 844, 848, 850
- Contouring Options, 843, 848, 850
 - Color Key, 850
- Control Areas, 591
- Controls, 351, 703
- Converge, 20, 668, 698
- Convergence, 20
- Convergence Tolerance, 668
- Converters, 398
- Converting Background Ellipses, 219
- Converting Background Lines, 218
- Converting Background Rectangles, 220
- Coordinates, 111
 - Screen, 111
- Copy, 20, 35, 300, 323, 529, 559
- Copy All, 529, 559
- Copy All menu, 529
- Copy Command, 35, 300, 323
- Copy Online, 84
- Copy Selection, 529, 559
- Copy Selection menu, 529
- Copy Window, 84
- Copying, 84
- Copying Simulator Data, 529, 559
- Cost, 144, 346, 350, 433, 612, 613, 615, 617, 713, 720
- Cost Curve, 144, 720
- Cost Information, 346, 350, 433
- Cost Models, 617
- Counter Flows, 857
- Create Equivalent, 285, 286

- Creating, 16, 1413
 - New Case, 16
 - New Project, 1413, 1414
 - Creating Injection Groups, 487
 - Criteria Dialog, 302
 - Cross Compound Units, 145
 - CTG, 743, 815, 816, 1469
 - Current Zoom Level, 308, 319
 - Curves, 353, 720
 - Custom, 581, 850
 - Custom Case Information Display, 581, 583
 - Custom Case Information Displays, 517, 582, 583, 584, 586, 587
 - Custom Color Map, 850
 - Custom Detail Dialog, 71
 - Custom Display, 583
 - Custom Expressions, 538, 548, 549, 787
 - Custom Expressions Display, 549
 - Custom Hint Values, 722
 - Custom Results Selection Dialog, 900
 - Customize, 68, 70, 71, 74
 - Cut, 35, 300, 323
- D**
- Data LimitSet, 1365
 - DC, 191, 396, 397, 398, 400, 591, 630, 631, 667, 678, 680
 - DC Transmission Line, 393, 394
 - Actual Flow, 394
 - Inverter Parameters, 393
 - DC Transmission Line Options, 389
 - DC Transmission Lines, 390, 392
 - Line Parameters, 390
 - Rectifier Parameters, 392
 - Decimal, 315, 526
 - Decimal places, 526
 - Decrease Decimals, 526
 - Default, 259, 302, 307, 308, 310, 316, 317, 318, 319
 - Background Color, 307
 - Drawing Values, 310
 - Default Drawing Options, 259
 - Default Drawing Options Dialog, 35, 36, 300, 301, 310, 316, 323, 324
 - Default Drawing Values, 310, 315
 - Default Values, 309
 - Define Expression, 538, 548
 - Define Fields/Strings, 582
 - Define Layers, 317
 - Define Layers button, 317
 - Defining, 750
 - Defining Contingencies, 750
 - Definitions, 743
 - Delete, 35, 36, 300, 301, 318, 323, 324
 - Object, 36, 301, 324
 - Deleting Injection Groups, 490
 - Delphi, 1144, 1152, 1161
 - Device Filtering, 543, 544, 546
 - Dialog, 302, 310
 - Difference, 730, 732, 733
 - Difference Case, 731, 732
 - Difference Flows, 730, 731, 732, 733, 735
 - Digits, 315
 - Right, 315
 - Directions, 867, 868
 - Disable Balancing, 668
 - Disable Checking Gen VAR Limits, 669
 - Disable Logging, 20
 - Disabled
 - Why is this field or line disabled, 518
 - Display, 36, 301, 319, 321, 324
 - Conditional, 319
 - Refresh, 527
 - Display Detail, 71
 - Display Grid Lines on Oneline, 305
 - Display Log, 20
 - Display Object, 35, 36, 37, 67, 71, 174, 178, 300, 301, 302, 311, 321, 323, 324
 - Display Object Properties, 321
 - Display Objects, 180, 182, 186, 191, 193, 198, 200, 202, 209, 222, 254, 255

Display Options, 81, 158, 307
 Display/Column Options, 526
 Display/Size, 37, 311, 313, 321
 Display/Size Tab, 313, 321
 Displays, 303, 308, 316, 507, 714
 Color Dialog, 316
 Distance, 708
 Document, 212
 Document Links on Onelines, 212
 Drawing Options, 259

E

Economic Dispatch Control, 701
 ED, 668, 701
 Edit, 17, 35, 36, 155, 300, 301, 302, 305, 310, 317, 318, 319, 323, 324
 Edit Menu, 35, 36, 300, 301, 323, 324
 Edit Mode, 16, 17, 19, 35, 36, 37, 39, 68, 154, 155, 159, 300, 301, 310, 311, 317, 318, 319, 323, 324, 328, 334, 338, 354, 361, 415, 421
 Edit Mode Introduction, 17
 Edit Mode Overview, 17, 154
 Edit Screen Layers, 318
 Edit Toolbar, 35, 300, 323
 Edit Tools, 35, 300, 323
 Enter new keyword here, 631
 Environment, 666, 685
 Equipment Mover, 254, 255
 Equivalencing, 285, 286
 Equivalencing Display, 262
 Equivalentents, 262, 285
 Equivalentents Display, 286
 Excel, 529, 559, 1181
 Export, 646, 705
 Extra Monitors, 1093
 Extra Width, 305

F

F1 key, 11
 Facility Analysis, 262, 293, 294, 296

Fault, 19, 348, 832, 836, 838, 841
 Submode, 19, 1320
 Fault Analysis, 831, 832, 835, 836, 837, 840, 841
 Field, 39, 302, 315, 331, 561
 Field Options, 471
 Fields, 159, 160, 170, 172, 175, 181, 183, 185, 195, 199, 201, 204, 221, 224, 225, 265, 331, 384, 386, 387, 411, 419, 561
 Key Fields, 561
 File, 128, 592, 1410
 File Management Options, 693
 File Menu, 128
 File options, 20
 Save Log, 20
 Fill Color, 313, 316
 Fill Options, 316
 Filter, 30, 265, 527, 531, 541, 542, 543, 544, 546, 547, 557, 674
 Find, 308, 523, 526, 553
 Find Branches that Create Islands, 721
 Find Dialog, 553
 Find Records, 523
 Finding Records, 523, 553
 Fixed Screen Size, 317
 Flow Sensitivities, 869
 Flow Visualization, 74
 Flowgates, 207, 208
 Flows, 70, 71, 74, 205, 869
 Font, 20, 37, 311, 313, 315, 317
 choosing, 315
 Font Properties, 315
 Font Tab, 37, 311, 313, 315
 Form Control, 518
 Format, 37, 119, 129, 141, 303, 311, 313, 315, 317, 318, 319, 321, 526, 755
 Format Field Properties, 320
 Format Menu, 318
 Format Multiple Objects, 119, 313
 Format Multiple Objects dialog, 317, 319, 321
 Levels/Layers Tab, 319

- Format Multiple Options dialog, 313
 - open, 313
- Format Selection Dialog, 303, 313
 - Font Tab, 37, 311
 - Levels/Layers Tab, 37, 311
 - Line/Fill Tab, 37, 311
- Format Toolbar, 37, 311
- Free Floating Window, 13, 20, 62, 116
- Front, 38, 312, 317
 - Bring, 317
- Fuel Cost, 615
- Fuel Type, 616
- Full CTG Solution, 1077
- Functional Description of Contour Options, 852
- Functions, 550
- Functions and Operators, 548, 549, 550

G

- Gauges, 39
- gcd, 143
- gcp, 142
- General, 681, 1284
- General Script Actions, 19, 1284
- Generation Chart, 710
- Generation Shift Factor, 880
- Generation Shift Factor Sensitivities, 880
- Generator, 71, 145, 255, 338, 340, 342, 343, 346, 348, 349, 350, 351, 353, 428, 430, 433, 435, 436, 610, 612, 613, 615, 617, 692, 703, 720, 836
 - Limits, 692
 - MW Limits, 692
 - Ramp Limits, 692
- Generator Capability Curves Format, 142
- Generator Cost, 610, 613
- Generator Cost Curve Data Options, 144
- Generator Cost Data Format, 143
- Generator Costs, 617
- Generator Cubic Cost, 615
- Generator Display Objects, 180
- Generator Economic Curves, 720
- Generator Field Information, 340
- Generator Fields, 181, 316, 340
- Generator Information, 342, 346, 349, 350, 428, 430, 433, 435, 436
 - Area, 349, 436
 - Costs, 346, 433
 - Owner, 349, 436
 - Zone, 349, 436
- Generator MW Marginal Cost, 615
- Generator Options, 338, 343, 348
 - Fault Parameters, 348
 - MW Control, 343
- Generator Piecewise Linear, 617
- Generator Reactive Power Capability Curve, 353
- Generator Records, 610, 836
- Generator Spinning Reserves, 592
- Generator/Load Cost Models, 612
- Generators, 81, 159, 180, 181, 254, 255, 277, 289, 340, 342, 350, 351, 353, 428, 431, 435, 486, 487, 490, 491, 493, 496, 497, 500, 591, 596, 606, 610, 612, 613, 615, 617, 637, 639, 641, 660, 692, 703, 720, 813, 836
 - Limits, 692
- Generic Model Fields, 224
- Geographic Data View, 572
 - Geographic Data View Styles, 579
 - Fields and Attributes, 574
 - General Display Options, 578
- Geography/Coordinates, 97
- Get Column Metrics, 540
- Get Parameters, 1143, 1144, 1146, 1148, 1150, 1152, 1153, 1155, 1158, 1205
- GetParametersMultipleElementFlatOutput Function, 1157
- Getting, 16
 - Started, 16
- GIS, 94, 95, 96, 98, 105, 107, 108, 110
 - Great Circle Distance Calculation, 95
 - Map Projections, 90
- GIS Shapefile Data, 99, 101, 102, 103

- Control, 99
- Format, 102
- Identify, 100
- Modify, 101
- Shape List, 103
- GIS Tools, 94, 95, 96, 105, 107, 108, 110
 - Closest Facilities to Point, 110
 - Delete All Measure Lines, 108
 - Export Oneline as Shapefile, 105
 - Great Circle Distance, 94
 - Insert Measure Line, 107
 - Populate Lon,Lat with Display X,Y, 96
 - Shape File Import, 98
- Global, 759
- Global Actions, 759, 783
- Governor, 738, 739
- Graph, 297, 709, 710, 711, 712
- Graphic Formats, 150
- Graphics, 151, 524
- Great Circle Distance Calculation, 95
- Great Circle Distance Dialog, 94
- Grid, 305, 540, 567
 - Snap Objects, 305
- Grid Metrics, 567
- Grid Metrics Dialog, 540
- Grid/Highlight Unlinked Objects, 70, 305
- Gridline Color, 305
- Gridlines, 305
- GSF, 880

H

- Help, 11
- High Zoom Level, 319
- Highlight Unlinked Objects, 305
- Highlighting, 305
 - Unlinked Objects, 305
- Hourly Summary, 890, 892
 - Local Menu, 892

I

- Identify Cross Compound Units, 145
- Images, 151, 524
- Immobile, 37, 311, 321
- Impedance, 366, 403, 404, 449, 839
- Impedance Correction Tables, 449
- Import, 646, 705
- Import PTI Subsystems Dialog, 495
- Include, 1123
- Include Suffix, 315
- Increase Decimals, 526
- Increase/Decrease Decimals, 526
- Indication, 39
- inf, 147
- Info/Interchange, 466
- Initialize, 1411
- inj, 146
- Injection Groups, 71, 146, 486, 487, 490, 491, 493, 495, 496, 498, 500, 592, 637, 639, 642, 1362
 - Import PTI Subsystems Dialog, 495
- Injection Groups Format, 146
- Input Page, 893
- Insert, 39, 160, 203, 271, 274, 275, 277, 278, 279, 280, 281, 318, 483
 - Area, 39
 - Area/Zone, 39
 - Background, 39
 - Directions, 868
 - Display Objects, 39
 - Fields, 39
 - Hyperlink, 39
 - Injection Group, 39
 - Interface, 39
 - Link, 39
 - Memo, 39
 - Picture, 39
 - Pie Charts, 39
 - Screen Layer, 318
 - Text, 39

- Zone, 39
 - Insert Palettes, 271
 - Install, 1122
 - Installing, 1122
 - InterArea, 205
 - Interchange, 597, 599
 - Interchange MW, 597, 599
 - Interface Data Format, 147
 - Interface Fields, 204, 481
 - Interface Information, 478, 480, 481, 482
 - Interface Limiters, 1108
 - Interfaces, 71, 147, 202, 203, 204, 206, 207, 280, 308, 456, 478, 480, 481, 482, 483, 484, 592, 634, 636, 773, 858, 1108, 1363
 - Introduction, 4, 926
 - Inverter Parameters, 393
 - IOA, 615
 - Island Based AGC, 676
 - Island Slack Bus, 644
 - Island-Based AGC, 667
 - Islands, 592, 644, 676, 737, 739
 - Iterated Linear Step, 1077, 1109
 - Iterations, 20, 668, 698
- J**
- Jacobian, 152, 662
- K**
- Key Field, 690
 - Key Fields, 561
 - Keyboard Short Cut Actions Dialog, 120
 - Keyboard Shortcuts, 120
- L**
- Labels, 456, 459
 - Layer, 37, 302, 311, 313, 317, 318, 319
 - Screen Stack, 38, 312
 - Stack, 37, 311
 - Layers, 114, 317, 319
 - choosing, 317
 - Level, 37, 114, 302, 311, 316, 317, 319
 - Background, 38, 312
 - Base, 38, 312
 - Middle, 38, 312, 317
 - Top, 38, 312, 317
 - Levels/Layers, 37, 311, 313, 317, 318, 319
 - Levels/Layers Options, 317, 318
 - Levels/Layers Tab, 37, 311, 313, 317, 319
 - Format Multiple Objects dialog, 319
 - Limit, 692, 1077
 - Limit Groups, 654, 692
 - Limit Monitoring, 777, 778, 780
 - Limit Monitoring Settings, 648, 649, 654, 692, 1077, 1110
 - Limit Monitoring Settings and Limit Violations Dialog, 649
 - Limit Violations, 692
 - Limiters, 1108
 - Line, 35, 36, 37, 71, 257, 275, 300, 301, 302, 311, 313, 316, 323, 324, 622, 624, 661, 837
 - Line Color, 316
 - Line Field Options, 384
 - Line Fields, 185, 316
 - Line Flow Gauge Options Dialog, 189
 - Line Flow Gauges, 188
 - Line Information, 382, 400
 - Line Loading Replicator, 724
 - Line Outage Distribution Factors, 861
 - Line Per Unit Impedance Calculator Dialog, 366
 - Line Shunt Display, 629
 - Line Shunts, 365
 - Line Shunts Information, 365
 - Line Taps, 256, 257
 - Line Thickness, 316
 - Line/Fill, 37, 311, 313, 316
 - Line/Fill Properties, 316
 - Line/Fill Tab, 37, 311, 313, 316
 - Line/Transformer Information, 439
 - Linear ATC, 1078, 1109
 - Linear Calculation Method, 1078

Linearized AC, 1078
 Lines, 773
 Lines Buses and Interfaces Tab -
 Contingencies Section, 774
 Lines/Transformers, 591, 837
 Links, 210, 212
 Other Onelines, 210
 List of Devices, 1159, 1161, 1163, 1165
 ListOfDevicesAsVariantStrings Function, 1167
 ListOfDevicesFlatOutput Function, 1168
 Load, 82, 354, 358, 360, 437, 612, 619, 789,
 813, 840
 Load Benefit Models, 620
 Load Chart, 710
 Load Display Objects, 182
 Load Field Information, 358
 Load Fields, 183, 316, 358, 789
 Load Flow, 21
 Load ID, 619
 Load Information, 437
 Load Modeling, 360
 Load Multiplier, 597, 599
 Load NERC Flowgates, 207
 Load Options, 354, 356, 357
 Load Information, 356
 OPF Load Dispatch, 357
 Load Schedules, 597, 599
 Load State, 1169, 1170
 Load Status, 619
 Load Throw Over, 789
 Load Variation, 597, 599
 Loading NERC Flowgates, 207
 Loads, 159, 182, 183, 254, 255, 278, 289, 354,
 358, 437, 456, 486, 487, 490, 491, 493, 496,
 497, 500, 591, 596, 612, 619, 620, 637, 639,
 641, 659, 813, 840
 Local Menu, 68, 518
 Local Menus, 587
 LODF, 861, 863, 865, 1077
 Log, 20, 27, 53
 Windows, 20

Loss, 871
 Sensitivities, 871
 Loss Sensitivities, 871
 Loss Sensitivity, 616
 Losses, 680, 711
 LTC Transformer, 379, 405, 443
 LTC Transformers, 407, 445

M

Maintain Fixed Screen Location, 317
 Maintain Fixed Screen Size, 317
 Make Default, 315
 Make Default button, 315
 Make-Up Power Sources, 813, 814
 Map Projections, 90
 Matrix Grids, 894
 Max Flow Min Cut, 296, 297
 Memo, 21, 470
 Memo Display, 21
 Memo Text, 223
 Menus, 128
 Merging Buses, 251
 Message Log, 20, 27, 53, 698
 Print, 20
 Save, 20
 Show, 20
 Using, 20
 Metrics, 540, 567
 Microsoft Excel, 529
 Middle, 38, 312, 317
 Level, 317
 Minimal, 71
 Minimal Show, 71
 Minimum Highlighted Object Pixel Size, 305
 Minimum Sensitivity, 668
 Mismatch, 608
 Model Conditions, 674, 1368
 Model Conditions Dialog, 556
 Model Conditions Display, 556
 Model Criteria, 674

Model Explorer, 504, 588
 Explore Pane, 504
 Fields Pane, 505
 Recent Tabs, 505
 Model Expressions, 786
 Model Fields, 224
 Model Filters, 557, 674, 1370
 Model Object, 67
 Modeling, 360, 556, 557, 612, 613, 617, 620, 737, 777, 783
 Models, 556, 613, 674
 Moderate Show, 71
 Modify, 16, 313, 316, 317, 318, 319, 321
 Orientation, 321
 Size, 321
 Mouse, 3, 13, 71
 Mouse Wheel Zooming, 71
 Movie Maker, 89
 Multiple Display Fields, 160
 Multiple Scenario, 1108
 Multiple Scenario Analysis, 1078, 1098
 Multiple Scenario ATC Analysis - Results Tab
 Local Menu Options, 1105
 Multiple Scenario ATC Dialog Results Tab, 1103
 Multiple Scenario ATC Dialog Scenarios Tab, 1100
 Multi-Section Lines, 192, 382, 596, 624, 1375
 Multi-Terminal, 396, 397, 398, 400, 631
 Mutual, 838, 839
 Mutual Impedance, 838, 839
 MVA flow, 73
 Mvar, 71, 407, 445
 Mvar Control, 407, 445
 MVR flow, 73
 MW, 71, 73, 707, 712
 MW Control, 343
 MW Distance, 707
 MW Marginal Cost, 615
 MW Transactions, 597, 599, 645, 712
 MW/MVR, 71

N

NERC, 207, 208
 Network, 39, 504
 Network Cut, 265, 266
 New, 136, 137
 New Case, 16, 17, 136
 Creating, 16
 New Online, 17, 137
 New Timepoint Dialog, 906
 Newton, 59, 60, 695, 696
 Newton-Raphson, 670, 698
 Nomogram Display, 636
 Nomogram Information, 484
 Nomogram Interface Limiters, 1108
 Nomograms, 484, 636

O

O&M, 615
 Objects, 21, 35, 36, 39, 67, 70, 71, 254, 264, 300, 301, 302, 308, 310, 313, 315, 316, 317, 318, 319, 321, 323, 324, 526, 1194
 Area, 308
 Area/Zone, 308
 Background Objects, 213, 215, 216, 217, 221, 222, 224, 316
 Bus, 308
 Display, 35, 36, 300, 301, 323, 324
 Find, 308
 Zone, 308
 Observing, 317
 online, 317
 Off-Nominal, 404
 Online, 35, 37, 39, 66, 67, 70, 71, 79, 81, 83, 111, 112, 119, 122, 134, 139, 140, 150, 158, 169, 170, 210, 268, 300, 302, 305, 307, 308, 311, 315, 317, 318, 319, 321, 323, 687, 714, 717
 Browsing Path, 688
 Conditional Display, 112, 119
 Coordinates, 687

- Diagrams, 66, 71, 122, 134, 158, 169, 170, 172, 174, 175, 181, 183, 186, 187, 260, 307
- Display, 70, 71, 81, 158, 305, 307
- Diagrams, 190
- Display Options, 687
- Displays, 714, 717
- Exporting, 150
- File, 685
- Font, 687
- Links, 72, 210
- Options, 71, 81, 119, 158, 307, 317, 319, 687
- Save, 687
- Screen Coordinates, 111
- Size, 687
- Text Fields, 221
- Visualizing, 687
- Online Display Options, 70, 97
 - Geography/Coordinates, 97
- Open, 20, 133, 134, 135, 302, 313, 317, 1416
 - Project, 1416
 - Screen Layers list, 317
- Open Case, 1171, 1172
- Operators, 550
- OPF, 357, 435, 469, 636, 702, 740, 986
 - Constraints, 988, 989, 990, 992, 993, 1000
 - Controls, 987, 998, 1002, 1013, 1029, 1030
 - Data, 994, 997, 1005
 - Example, 996, 1031, 1034, 1035, 1037
 - Load Dispatch Options, 357
 - LP OPF, 995, 1008, 1009, 1010
 - Options, 997, 998, 1000, 1002, 1005, 1008, 1009, 1010, 1011, 1012, 1013, 1014
 - Records, 994, 1015, 1017, 1018, 1021, 1022, 1023, 1025, 1027
- OPF Control, 702
- OPF DC Lines Records, 1024
- OPF Options Advanced Options, 1004
- OPF Options and Results, 997
- OPF Reserves
 - Case Information Display, 1072
 - Constraints, 1068
 - Controls, 1066
 - Example, 1073
 - Objective Function, 1070
 - Overview, 1064
 - Reserve Requirement Curves, 1069
 - Reserves Bids, 1067
 - Results and Pricing, 1071
 - Topics, 1065
- Optimal Power Flow, 435, 469, 636, 702, 740, 986
 - Constraints, 988, 989, 990, 992, 993, 1000
 - Controls, 987, 998, 1002, 1013, 1029, 1030
 - Data, 994, 997, 1005
 - Example, 996, 1031, 1034, 1035, 1037
 - LP OPF, 995, 1008, 1009, 1010
 - Options, 997, 998, 1000, 1002, 1005, 1008, 1009, 1010, 1011, 1012, 1013, 1014
 - Records, 994, 1015, 1017, 1018, 1021, 1022, 1023, 1025, 1027
 - Options, 20, 71, 158, 259, 302, 305, 307, 308, 310, 313, 316, 318, 319, 526, 666, 667, 668, 676, 678, 681, 682, 685, 689, 699, 708, 737
 - Options/Tools, 70, 83, 307, 310, 709, 735, 869
 - Orientation, 313, 321
 - Orientation Setting, 321
 - OTDF, 1093
 - Other Applications, 559
 - Other Display Object Properties, 321
 - Out of Service Elements, 687
 - Outages, 596
 - Out-of-Service Elements, 72
 - Visualizing, 72
 - Overheated Lines, 692
 - Overview, 66, 154, 270, 293, 486, 496, 927, 986, 1039, 1410
 - Owned, 659, 660, 661
 - Owned Bus Records, 658
 - Owned Generator Records, 660
 - Owned Line Records, 661
 - Owned Load Records, 659

Owner, 349, 436, 541, 656, 657, 658, 659, 660, 661, 1378
 Owner Display Objects, 167
 Owner Display Options Dialog, 168
 Owner Fields on Onelines, 173
 Owners, 541

P

Palettes, 270, 271
 Pan, 3, 50, 112, 308, 317
 Zoom, 112
 Pan/Zoom, 113, 308
 New Location, 309
 Pan/Zoom Control, 113, 308
 Panning, 308
 Participation Factor Control, 701
 Participation Factors, 351, 611, 701, 703, 739
 Participation Points, 496, 497, 500, 641
 Paste, 35, 36, 300, 301, 323, 324, 529, 530, 559
 Paste Buffer, 35, 36, 300, 301, 323, 324
 Paste Format Dialog, 314
 Path, 72, 333
 Browsing, 72
 Pathname, 592
 Pause, 27, 53, 89
 Per Unit, 366, 403
 Phase, 404, 409, 447, 626
 Phase Shift, 404, 409, 447
 Pie Chart / Gauge Dialogs, 231
 Pie Chart / Gauge Example, 242
 Pie Chart / Gauge Style Dialog, 234
 Pie Chart / Gauge Style Dialog - Gauge Parameters Tab, 240
 Pie Chart / Gauge Style Dialog - Open Parameters Tab, 238
 Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab, 239
 Pie Chart / Gauge Style Dialog - Standard Parameters Tab, 236
 Pie Chart Relative, 73

Pie Chart Style, 73
 Pie Charts, 70, 73, 187, 206, 482
 Pie Charts/Gauges General Options, 230
 Pie Charts/Gauges Interfaces, 228
 Pie Charts/Gauges Lines, 226
 Pies, 39
 Populate Lon,Lat with Display X,Y, 96
 Popup menu, 318
 Post Power Flow, 668, 674
 Post-contingency, 1110
 Potential Misplacements Dialog, 255
 Power Control, 430
 Power Flow, 20, 593, 595, 608, 667, 668, 680, 698, 739, 789, 856
 Submode, 1312
 Visualize, 856
 Power Flow List, 593, 595
 Power Flow Solution, 3, 20, 646, 666, 667, 668, 674, 676, 678, 680, 681, 682, 685, 691, 692, 698, 701, 705, 707, 708, 709, 710, 711, 712, 713, 714, 717, 737, 739, 789, 861
 Power Flow Solution Advanced Options, 671
 Power Flow Solution Message Log, 683
 Power Flow Solution Actions, 668
 Power System Model, 35, 36, 67, 300, 301, 323, 324, 360, 456, 556, 557, 612, 613, 617, 620
 Power System Scaling, 289
 Power Technologies, 754
 Power Technology, 755
 Power Transfer Distribution Factors, 856, 857, 1077
 PowerFlow, 19
 Submode, 19, 1312
 PowerPoint, 84
 PowerWorld, 11, 16, 1194
 PowerWorld Corporation, 11
 PowerWorld Project, 1410, 1411, 1412, 1413, 1414, 1416
 PowerWorld Simulator, 11, 16
 Present Case, 730, 731, 732, 733, 735
 Present Topological Differences, 735

- Presentations, 84
 - Pre-Transfer Value Estimate, 1110
 - Prevent Controller Oscillations, 670
 - Print, 122, 123, 125
 - Print Message Log, 20
 - Print Options, 123
 - Print Title Bar, 123
 - Print Window, 123
 - Printer Setup, 125
 - Printing, 122, 123, 125
 - ProcessAuxFile, 1173, 1174
 - Project, 1410, 1411, 1412, 1413, 1414, 1416
 - Properties, 305, 315, 316, 321
 - Display Object, 321
 - PSS/E, 754, 755, 757
 - PSS/E Load Throw Over Files, 756
 - PTDF, 3, 486, 707, 856, 857, 861, 866, 867, 868, 874, 1077, 1093, 1110
 - Directions, 867
 - Publish, 151, 524
 - PV, 927, 928, 929, 946, 952, 955, 958, 960, 981, 983
 - Options, 931, 932, 935, 943, 944, 949, 977, 984
 - Submode, 19, 1321
- Q**
- Quick Access Toolbar, 26
 - Quick Power Flow List, 537, 558, 593, 595
 - QV, 958, 960, 961, 963, 966, 968, 970, 978, 979, 981, 983
 - Options, 943, 949, 965, 977, 984
 - Submode, 19, 1323
- R**
- Range, 558
 - Range of Numbers, 558
 - Reactive Power Capability, 353
 - Recently Opened Cases, 135
 - Records, 67, 522, 523, 529
 - Sorting, 518, 522, 527
 - Rectangle Selector, 313
 - Rectangular Selector, 313
 - Rectifier Parameters, 392
 - Reference State, 745
 - Refresh, 62, 71, 116, 245, 527
 - Refresh Anchors, 245
 - Refresh Display, 527
 - Regulation, 610
 - Relationship Between Display Objects, 35, 36, 300, 301, 323, 324
 - Relinking, 305
 - Remote Regulation, 606, 610
 - Remote Regulation Viability, 698
 - Remotely Regulated, 606, 610
 - Renumber Areas/Zones/Substations Dialog, 250
 - Renumbering, 246, 247, 249
 - Report Violations, 780
 - Report Writing, 777, 790
 - RequestBuildDate Property, 1193
 - Required Field, 527, 690
 - Required Fields, 563
 - Resize, 37, 311, 313, 317
 - Resizing, 317
 - Restore, 682
 - Restore Default Values, 309
 - Restore Last Solution, 682
 - Restore State, 682
 - Result Tab, 1108
 - Results, 895
 - Constraints, 895
 - Results Grid Pages, 901
 - Ribbon, 23
 - Case Information, 30
 - Draw, 34, 157
 - Clipboard, 35, 300, 323
 - Formatting, 314
 - Individual Insert, 39
 - Quick Insert, 41
 - Select, 43, 304

- Onelines
 - Active, 46
 - Onelines, 44
 - Onelines
 - Zoom, 50
 - Options, 51
 - Quick Access Toolbar, 26
 - Tools, 53
 - Tools
 - Edit Mode, 55
 - Tools
 - Other Tools, 56
 - Tools
 - Run Mode, 58
 - Window, 62, 116
 - Right-clicking, 318
 - online, 318
 - Run, 18, 20
 - Run Mode, 18, 19, 72, 423, 426, 428, 437, 439, 450, 454, 856, 1304
 - Difference Flows, 731, 732, 733, 735
 - Fault Analysis, 831, 832, 835, 836, 837, 840, 841
 - Governor Power Flow, 737, 738, 739
 - Introduction, 18
 - LODF, 861, 863
 - Options, 738
 - Script, 1304
 - Sensitivities, 869, 871, 880
 - Tools, 720, 737, 740, 759, 786, 813, 856, 861, 874, 880
 - Tools & Options, 89, 720, 731, 732, 733, 735, 737, 739, 740, 744, 745, 751, 754, 755, 757, 759, 760, 762, 763, 764, 766, 769, 771, 773, 775, 777, 778, 780, 783, 786, 789, 790, 792, 793, 794, 796, 799, 802, 813, 814, 831, 832, 835, 836, 837, 838, 839, 840, 841, 856, 857, 861, 863, 865, 866, 867, 868, 869, 871, 874, 875, 878, 880
 - RunScriptCommand, 1175, 1176
- S**
- Save, 20, 50, 114, 138, 139, 152, 560, 1177, 1179
 - Save Case, 560, 1177
 - Save Case Information Data, 560
 - Save Hidden Layers, 115
 - Save Log, 20
 - Save View, 50, 114
 - Save/Edit/Delete, 114
 - Saving NERC Flowgates, 208
 - Scaling, 289, 558
 - Schedules, 597, 599
 - SCOPF, 435, 469, 636, 701, 740, 1039, 1040, 1041, 1042, 1043, 1048, 1050, 1052, 1053, 1057
 - Constraints, 1044, 1045, 1046
 - Controls, 1048, 1058
 - Example, 1059, 1062, 1063
 - LP, 1052, 1055, 1056
 - LP OPF, 995, 1008, 1009, 1010
 - OPF Constraints, 988, 989, 990, 992, 993, 1000
 - OPF Controls, 987, 998, 1002, 1013, 1029, 1030
 - OPF Data, 994, 997, 1005
 - OPF Example, 996, 1031, 1034, 1035, 1037
 - OPF Options, 997, 998, 1000, 1002, 1005, 1008, 1009, 1010, 1011, 1012, 1013, 1014
 - OPF Records, 994, 1015, 1017, 1018, 1021, 1022, 1023, 1025, 1027
 - Screen Coordinates, 111
 - Screen Layer Options, 319
 - Screen Layer Options dialog, 318
 - Screen Layers, 318, 319
 - Screen Layers list, 317
 - open, 317
 - Screen Location, 308, 317
 - Screen Size, 317
 - Fixed, 317
 - Script Command Execution Dialog, 19
 - Script Command Execution Window, 19

Script Commands, 19
 Script General Actions, 19, 1284
 Script Mode, 19, 27, 53, 1278, 1280, 1282, 1297, 1325, 1329
 Actions, 1284, 1297, 1299, 1304, 1312, 1316, 1318, 1320, 1321, 1323
 Data, 1336, 1337
 Data, 1325, 1326, 1329, 1330, 1331, 1332, 1333, 1334
 Data, 1338
 Data, 1339
 Data, 1340
 Data, 1341
 Data, 1342
 Data, 1348
 Data, 1349
 Data, 1350
 Data, 1351
 Data, 1352
 Data, 1353
 Data, 1355
 Data, 1357
 Data, 1358
 Data, 1360
 Data, 1361
 Data, 1362
 Data, 1363
 Data, 1365
 Data, 1366
 Data, 1367
 Data, 1368
 Data, 1369
 Data, 1370
 Data, 1371
 Data, 1375
 Data, 1376
 Data, 1377
 Data, 1378
 Data, 1381
 Data, 1382
 Data, 1383
 Data, 1384
 Data, 1385
 Data, 1386
 Data, 1387
 Data, 1388
 Data, 1389
 Data, 1391
 Data, 1392
 Data, 1393
 Data, 1394
 Data, 1395
 Data for Display Auxiliary Files, 1380, 1396, 1397, 1398, 1400, 1401, 1402, 1403, 1404, 1405, 1406, 1407
 Files, 1278, 1281, 1282, 1325
 Script, 1280, 1282, 1284, 1297, 1299, 1304, 1312, 1316, 1318, 1320, 1321, 1323
 Script Mode Introduction, 19
 Search, 505, 526, 555
 Security Constrained OPF, 435, 469, 636, 701, 740, 1039, 1040, 1041, 1042, 1043, 1048, 1050, 1052, 1053, 1057
 Constraints, 1044, 1045, 1046
 Controls, 1048, 1058
 Example, 1059, 1062, 1063
 LP, 1052, 1055, 1056
 LP OPF, 995, 1008, 1009, 1010
 OPF Constraints, 988, 989, 990, 992, 993, 1000
 OPF Controls, 987, 998, 1002, 1013, 1029, 1030
 OPF Data, 994, 997, 1005
 OPF Example, 996, 1031, 1034, 1035, 1037
 OPF Options, 997, 998, 1000, 1002, 1005, 1008, 1009, 1010, 1011, 1012, 1013, 1014
 OPF Records, 994, 1015, 1017, 1018, 1021, 1022, 1023, 1025, 1027
 Select, 36, 301, 302, 316, 317, 321, 324
 Select By Criteria, 302, 313
 Select OK, 316
 Selected group, 303
 Selected objects, 265, 302

- Selecting Format, 316
- Seller, 1078, 1093
- Send, 38, 312, 317
 - Back, 38, 312, 317
- Send All to Excel, 530
- Send to Excel, 530
- Sensitivities, 764, 863, 869, 871, 875, 880
- Sensitivity Calculations, 765
- Sequence Data Format, 148
- Series Capacitor, 198, 381, 386
 - Fields, 386
- Series Capacitor Fields, 199
- Set Custom Detail, 71
- Set Generator Participation Factors, 351, 703
- Setting, 307
 - Background Color, 307
- Setting Default Drawing Options, 259
- Shape File Import, 98
- Shapefile, 99, 100, 101, 102, 103
 - Database Record Dialog, 104
 - Export Online as Shapefile, 105
 - GIS Shapefile Data Control, 99
 - GIS Shapefile Data Format, 102
 - GIS Shapefile Data Identify, 100
 - GIS Shapefile Data Modify, 101
 - GIS Shapefile Data Shape List, 103
- Shortcuts, 120
- Shortest Path Between Elements, 333
- Show Animated Flows, 74
- Show Fields Primary, 584
- Show Fields Secondary, 586
- Show Thumbnail View, 77
- Shunts, 159, 186, 200, 254, 255, 279, 289, 365, 415, 450, 632, 841
- SimAuto, 1121, 1122, 1123, 1124, 1126, 1127, 1128, 1130, 1133, 1134, 1136, 1137, 1167, 1168, 1186, 1194
 - ChangeParametersMultipleElement, 1133
 - ChangeParametersMultipleElementFlatInput, 1136
 - ChangeParametersSingleElement, 1130
- Delphi, 1144, 1152, 1161
- Functions, 1128, 1129, 1139, 1141, 1143, 1150, 1158, 1159, 1169, 1171, 1173, 1175, 1177, 1179, 1181, 1183
- Get Parameters, 1143, 1144, 1146, 1148, 1150, 1152, 1153, 1155, 1158, 1205
- List of Devices, 1159, 1161, 1163, 1165
- ListOfDevicesAsVariantStrings, 1167
- ListOfDevicesFlatOutput, 1168
- Load State, 1169, 1170
- MATLAB, 1146, 1153, 1163
- Open Case, 1171, 1172
- ProcessAuxFile, 1173, 1174
- Properties, 1186, 1187, 1189, 1191
- RunScriptCommand, 1175, 1176
- Sample Code, 1131, 1134, 1137, 1140, 1142, 1144, 1146, 1148, 1152, 1153, 1155, 1161, 1163, 1165, 1170, 1172, 1174, 1176, 1178, 1180, 1182, 1184, 1188, 1190, 1192
- VB, 1148, 1155, 1165
- Version 9.0, 1195, 1196, 1197, 1198, 1200, 1201, 1202, 1203, 1204, 1205, 1206, 1207, 1208, 1209, 1210, 1211, 1212, 1213, 1214
- Simulation, 20, 133
- Simulation Control, 27, 53, 59, 695
- Simulator Options, 666
- Single Linear Step, 1077
- Single Solution, 59, 60, 668, 695, 696
- Size, 71, 313, 315, 317, 321, 526
 - modify, 321
- Size/color, 316
- Slack, 592
- Slack Bus, 329, 592, 671
- Slide Presentations, 84
- Snap Objects, 305
 - Grid, 305
- Solution, 20, 608, 646, 666, 667, 668, 674, 676, 678, 680, 681, 682, 685, 691, 692, 698, 702, 705, 707, 708, 709, 710, 711, 712, 713, 716, 717, 744
- Solution Options, 666, 667, 668, 676, 678, 681, 682, 685, 689, 698, 708, 737, 739, 744
- Solving, 698

Sort, 522, 527
 Sorting, 518, 522, 527
 Records, 529
 Sorting Records, 522
 Special Data Sections, 1402
 Specifying and Maintaining a List of Timepoints, 905
 Spinning Reserves, 592
 Splitting Buses, 252, 253
 Stability, 927
 Stack, 37, 311, 313, 317
 Stack Level, 37, 311, 313, 317
 Status Bar, 22
 Storage, 667, 682
 Submenu, 69
 Submode, 19
 Submodes, 19
 ATC, 19
 Case, 19
 Contingency, 19
 Fault, 19
 PowerFlow, 19
 PV, 19
 QV, 19
 Substation, 70, 71, 81, 179, 308, 336, 609, 717
 Field Options, 336
 Fields, 179
 Object, 308
 Substation Display, 81, 717
 Substation Display Options, 81
 Substation Field Options, 336
 Substation Fields on Onelines, 179
 Substation Information, 334, 426
 Substation Records, 609
 Substation View, 537, 717
 Substations, 178, 281, 334, 426, 717
 Super Area, 172, 473, 476, 603
 Super Area Display Objects, 165
 Super Area Display Options Dialog, 166
 Super Area Information, 473, 476

Supplemental Classification, 87
 Supplemental Data, 87
 Creating Records, 87
 Fields, 225
 Linking Display Objects, 88
 Switched Shunt, 82, 200, 415, 419, 450, 632, 841
 Switched Shunt Fields, 201, 316, 419
 Switched Shunt Information, 415, 419, 450, 841
 Switched Shunts, 201, 254, 255, 279, 415, 419, 450, 456, 591, 596, 841
 System Scaling, 289

T

Tap, 256, 626
 Tapping Transmission Lines, 256
 Taps, 257
 Terms, 111
 x-y, 111
 Text, 71, 221, 222, 316, 317, 526, 555
 Text Fields, 221
 Text/font, 313
 Three Winding Transformers, 196, 413, 628
 Throw Over, 789
 Thumbnail, 70, 77
 Background, 78
 Size, 78
 Thumbnail View, 70, 77
 Tie Lines, 468
 Tie Lines between Areas, 600
 Tie Lines between Zones Display, 602
 Time Step Simulation, 882, 883, 885, 888, 889, 890, 893, 894, 895, 898, 899, 900, 901, 902, 905, 906, 907, 908, 909, 911, 912, 913, 915, 916, 918, 920, 921, 924
 Binding Constraint Dialog, 898
 Change Timepoint Time Dialog, 907
 Custom Results Selection Dialog, 900
 Dialog, 885
 Hourly Summary, 890

- Input Page, 893
- Insert/Scale Column Dialog, 909
- Loading Hourly Input Data, 911
- Matrix Grids, 894
- New Timepoint Dialog, 906
- Options, 902
- Pages, 889
- Quick Start, 883
- Results, 899
- Results Grid Pages, 901
- Results: Constraints, 895
- Running a Timed Simulation, 920
- Running OPF and SCOPF, 921
- Schedule Dialog, 913
- Schedule Subscription Dialog, 916
- Schedule Subscriptions Page, 918
- Schedules Page, 915
- Setting up Scheduled Input Data, 912
- Specifying and Maintaining a List of Timepoints, 905
- Storing Input Data and Results, 924
- Toolbar, 888
- TSB Case Description Page, 908
- TLR, 874, 875, 878, 880
- TLR Multiple Device Type, 878
- TLR sensitivities, 874, 875
- TLR Sensitivities, 880
- Toggle Full Screen, 69
- Tools, 307, 310
- Top, 38, 312, 317
 - level, 317
- Topological Differences, 735
- Topology Processing, 1215, 1217, 1218, 1219, 1220, 1221, 1222, 1223, 1224, 1226, 1228, 1229, 1230, 1231, 1232, 1233, 1234, 1235, 1237
- Total Digits, 315
- Transactions, 597, 599, 645, 646, 705, 712
- Transfer, 1098, 1108
- Transfer Limit, 1108
- Transfer Limit For Scenario, 1098
- Transfer Limiters, 1098, 1108
- Transfer Limiters Display, 1108
- Transfer Result Reporting Options, 1085
- Transformer AVR Dialog, 379, 405, 443
- Transformer Control, 377, 401
- Transformer Fields, 195, 316
- Transformer Impedance Correction, 404
- Transformer Impedance Correction Tables, 449
- Transformer Options, 361
- Transformer Phase Shifting, 409, 447
- Transformers, 159, 186, 193, 195, 196, 254, 255, 361, 365, 377, 379, 387, 401, 403, 404, 405, 407, 409, 411, 413, 439, 443, 445, 447, 449, 591, 622, 624, 626, 628, 654, 661, 692, 837
 - Control, 377, 401
 - Fields, 387, 411
- Transient Stability, 1238, 1239, 1241, 1242, 1244, 1245, 1247, 1249, 1252, 1256, 1260, 1262, 1264, 1266, 1268, 1269, 1271, 1273, 1275
- Transmission Line, 393, 394
 - DC Actual Flow, 394
 - DC Inverter Parameters, 393
- Transmission Line Information, 837
- Transmission Line Options, 361
- Transmission Line Parameter Calculator, 367
- Transmission Line Transformer Options Display, 363
- Transmission Line/Transformer Options, 364, 377, 381, 401
 - Parameters/Display, 364
 - Series Capacitor, 381
 - Transformer Control, 377, 401
- Transmission Lines, 184, 185, 191, 192, 193, 254, 255, 275, 361, 365, 384, 389, 390, 392, 439, 591, 622, 624, 630, 654, 661, 692, 837
 - DC, 389
 - DC Line Parameters, 390
 - DC Rectifier Parameters, 392
 - Fields, 384
- Transmission Loading Relief, 874, 875, 878, 880

Transmission Loading Relief Sensitivities, 874, 875, 880

Turns Ratio, 404

Tutorial, 16, 815, 816, 1469

Contingency Analysis - Page 10, 829

Contingency Analysis - Page 11, 830

Contingency Analysis - Page 4, 821

Contingency Analysis - Page 5, 823

Contingency Analysis - Page 6, 824

Contingency Analysis - Page 7, 826

Contingency Analysis - Page 8, 827

Contingency Analysis - Page 9, 828

Tutorial Contingency Analysis, 815, 1469

Tutorial Creating a New Case, 1418

Tutorial Solving an OPF, 1462

Tutorial Starting with an Existing Case, 1447

Tutorial Contingency Analysis, 816

Tutorials, 815, 1418, 1419, 1421, 1423, 1424, 1427, 1429, 1431, 1433, 1436, 1438, 1442, 1444, 1447, 1448, 1449, 1450, 1451, 1452, 1453, 1454, 1455, 1456, 1457, 1458, 1459, 1460, 1461, 1462, 1463, 1464, 1465, 1466, 1467, 1469

U

Uncheck, 321

Anchored, 321

Undo, 325

Unit Type, 616

Unlinked Display Objects, 264

Unlinked Objects, 264, 305

Highlighting, 305

Unused Bus Numbers, 269

Use, 321

Display/Size Tab, 321

Use Absolute Values, 71

Mvar, 71

MW, 71

Use Area/Zone/Owner Filters, 30, 527, 531

Use Background Fill, 316

Use Conditional Display, 319

User-Defined Case Information, 504, 508, 588

Using Difference Flows, 733

Using the Online Alignment Grid, 306

V

Validate, 20

Variable O&M, 615

View, 50, 70, 77, 114, 158, 210, 714

View Name, 114

Violations, 780

Visualize, 856, 858

Power Flows, 856, 858

PTDFs, 856

Visualizing, 72, 687

Out-of-Service Elements, 72

Voltage Adequacy, 927

Voltage Control, 430

Voltage Gauges, 176, 177

Voltage Sensitivities, 869

Voltage Stability, 927

W

Warning, 73

Warning Color, 73

Warning Percent, 73

Default values, 73

Warning Size Scalar field, 73

Web, 11

Web Publishing, 151, 524

Weighted Transmission Loading Relief, 879

Why is this field or line disabled, 518

Wind Control Mode, 343

Window, 13, 20, 529

Windows Basics, 13

Word, 84

Word Processor, 84

X

X Grid Spacing, 305

Index

XF, 626

x-y, 111, 114

Y

Y Grid Spacing, 305

Ybus, 152, 663

Z

Zero Sequence, 838

Zone, 170, 302, 308, 349, 421, 436, 454, 541

 Object, 308

Zone Display Objects, 163

Zone Display Options Dialog, 164

Zone Fields, 170

Zone Information, 421, 454

Zones, 456, 541, 592, 634

Zoom, 37, 50, 71, 73, 77, 112, 302, 305, 308,
311, 317, 318, 319

 Area, 50

 In, 50

 Levels, 50, 73, 77, 112, 114, 302, 305, 317,
318, 319

 Out, 50

 Pan, 112

 Percentage, 73

 Toolbar, 50

Zoom/Pan/Find, 112, 308, 317