

# Table of Contents

<b>Chapter 1: Getting Started With PowerWorld</b>	<b>15</b>
Introduction to PowerWorld Simulator .....	15
What's New in Version 8.0 .....	16
About this Manual .....	17
Help .....	17
PowerWorld Interface .....	18
Windows Basics .....	18
PowerWorld Simulator: Getting Started .....	18
Edit Mode Introduction .....	18
Run Mode Introduction .....	18
Message Log .....	19
Web Publishing .....	19
Toolbars .....	20
Using the Toolbars .....	20
Program Palette .....	20
File Palette .....	21
Edit Palette .....	21
Insert Palette .....	21
Format Palette .....	21
List Display Palette .....	22
Zoom Palette .....	22
Pan/Zoom Palette .....	22
Options/Info Palette .....	22
Run Mode Palette .....	22
<b>Chapter 2: Using Online Diagrams</b>	<b>23</b>
Online Diagram Overview .....	23
Relationship Between Display Objects and the Power System Model .....	23
Online Tools and Options .....	24
Online Local Menu .....	24
Online Display Options Dialog .....	25
Display Options .....	26
Pie Chart Options .....	27
Animated Flows Options .....	28
Thumbnail View .....	29
Set Custom Detail Dialog .....	30
Online Animation .....	30
Copying Onelines to Other Programs .....	31
Changing the Online View .....	31
Online Screen Coordinates .....	31
Online Zooming and Panning .....	31
Save View Level Dialog .....	32
Window Menu Command .....	33
Online Conditional Display of Objects .....	34
Printing Online Diagrams .....	34
Printing Online Diagrams .....	34
Print Options Dialog .....	34
<b>Chapter 3: Creating, Loading, and Saving Simulator Cases</b>	<b>37</b>
File Menu .....	37
Case Formats .....	37

Validating a Case.....	38
Printer Setup.....	39
Opening Files.....	39
Opening a Simulation Case .....	39
Opening a Oneline Diagram.....	39
Recently Opened Cases .....	40
Creating Files.....	40
Building a New Case .....	40
Building a New Oneline.....	40
Closing Files .....	40
Saving Cases .....	40
Saving a Oneline .....	41
Close Oneline.....	41
Auxiliary Files.....	41
Save/Load Auxiliary Data.....	41
Area/Zone Load Schedule Format (*.shd) .....	41
Automatic Area Controls Format (*.aac).....	42
Auxiliary File Format (*.aux).....	42
Auxiliary Data File Type .....	43
Generator Capability Curves Format (*.gcp).....	44
Generator Cost Data Format (*.gcd).....	44
Generator Cost Curve Data Options.....	45
Identify Cross Compound Units.....	45
Injection Groups Format (*.inj) .....	46
Interface Data Format (*.inf).....	46
Sequence Data Format.....	47
Exporting Onelines in Different Graphic Formats.....	49
Saving Images as Jpegs.....	49
Saving Admittance Matrix and Jacobian Information .....	50
Templates.....	50
Using Templates.....	50
Case Templates .....	51
Oneline Templates.....	51

## **Chapter 4: Building a Oneline Diagram 53**

Overview .....	53
Edit Mode Overview.....	53
Edit Mode General Procedures.....	53
Insert Menu .....	54
Oneline Display Options .....	55
Anchored Objects .....	55
Inserting and Placing Multiple Display Fields.....	55
Areas and Zones.....	56
Area/Zone/Super Area Display Objects.....	56
Area/Zone/Super Area Display Options Dialog.....	56
Area Fields on Onelines.....	57
Zone Fields on Onelines .....	57
Buses .....	59
Bus Display Objects.....	59
Bus Fields on Onelines .....	60
Substations.....	61
Substation Display Objects .....	61
Generators.....	61
Generator Display Objects.....	61
Loads .....	62
Load Display Objects.....	62

Transmission Lines .....	63
Transmission Line Display Objects .....	63
Line Fields on Onelines .....	64
Circuit Breakers on Onelines.....	64
Line Flow Pie Charts on Onelines .....	65
DC Transmission Line Display Objects .....	65
Transformers .....	66
Transformer Display Objects.....	66
Transformer Fields on Onelines .....	67
Series Capacitors.....	68
Series Capacitor Display Objects .....	68
Series Capacitor Fields on Onelines .....	69
Switched Shunts .....	69
Switched Shunt Display Objects.....	69
Interfaces.....	70
Interface Display Objects .....	70
Inserting Interfaces Dialog .....	71
Interface Fields on Onelines .....	72
InterArea Flow Options Dialog .....	72
Interface Pie Charts on Onelines.....	72
Loading NERC Flowgates.....	73
Saving NERC Flowgates.....	73
Online Links.....	73
Links to Other Onelines .....	73
Document Links on Onelines .....	74
Background Objects and Text .....	74
Background Lines on Onelines .....	74
Background Rectangles on Onelines .....	75
Background Ellipses on Onelines.....	75
Background Pictures on Onelines .....	75
Text on Onelines .....	76
<b>Chapter 5: Edit Mode Tools and Options</b> .....	<b>77</b>
General Tools .....	77
Appending a Case .....	77
Bus Renumbering Dialog .....	77
Bus Renumbering: Automatic Setup of Bus List Options .....	78
Bus Renumbering: Bus Change Options .....	79
Splitting Buses.....	79
Split Bus Dialog.....	80
Equipment Mover.....	81
Potential Misplacements Dialog .....	82
Tapping Transmission Lines .....	82
Automatic Line Tap Dialog .....	82
Setting Default Drawing Options .....	83
KV-Based Color Coding.....	85
Bus Selection Page .....	86
List Unlinked Display Objects.....	87
List All Display Objects .....	88
Insertion Using Palettes .....	88
Palette Overview .....	88
Using the Insert Palettes.....	88
Automatic Object Insertion .....	90
Automatically Inserting Buses .....	90
Automatically Inserting Transmission Lines .....	90
Automatically Inserting Generators .....	91

Automatically Inserting Loads .....	92
Automatically Inserting Switched Shunts .....	93
Automatically Inserting Interfaces.....	93
Automatically Inserting Substations.....	94
Automatically Inserting Borders.....	95
Equivalencing .....	96
Equivalents.....	96
Equivalents Display .....	96
System Scaling.....	98
Scaling .....	98
<b>Chapter 6: Editing Online Diagrams</b>	<b>101</b>
Overview .....	101
Edit Menu .....	101
Selecting Objects.....	102
Rectangle Selector .....	102
Select by Criteria Dialog .....	102
Changing Basic Display Object Properties.....	103
Drawing Grid .....	103
Setting Background Color .....	103
Zoom, Pan, and Find .....	103
Default Drawing Values .....	104
Format Menu .....	104
Format Selection Dialog.....	105
Font Properties.....	106
Line/Fill Properties.....	106
Other Display Object Properties.....	107
Cutting and Pasting.....	107
Cut Command .....	107
Copy Command.....	107
Paste Command.....	108
Delete Command.....	108
<b>Chapter 7: Properties of Simulator Objects</b>	<b>109</b>
Edit Mode Properties and Information .....	109
Zone Properties .....	109
Zone Information (Edit Mode) .....	109
Bus Properties.....	109
Bus Information (Edit Mode) .....	109
Bus Field Information.....	111
Shortest Path Between Buses .....	113
Substation Properties.....	113
Substation Information (Edit Mode).....	113
Generator Properties .....	114
Generator Information (Edit Mode).....	114
Generator Information: Display .....	115
Generator Information: Cost Data .....	115
Generator Information: MW Control .....	116
Generator Information: Voltage Control.....	117
Generator Information: Fault Parameters .....	118
Generator Cost Information .....	118
Generator Participation Factors Information.....	119
Generator Reactive Power Capability Curve .....	120
Load Properties .....	120
Load Information (Edit Mode) .....	120
Load Modeling.....	122

Line Properties .....	123
Line/Transformer Information (Edit Mode).....	123
Line Shunts Information.....	125
Series Capacitor Information .....	125
Multi-Section Line Information.....	125
Line Field Information .....	127
Series Capacitor Fields Information Dialog .....	128
DC Line Information.....	128
DC Transmission Line Information .....	128
DC Line Information: Line Parameters .....	128
DC Line Information: Rectifier Parameters .....	129
DC Line Information: Inverter Parameters .....	130
DC Line Information: Actual Flows .....	130
Multi-Terminal DC Line Information.....	130
Multi-Terminal DC Record Information .....	130
Multi-Terminal DC Bus Information .....	131
Multi-Terminal DC Converter Information .....	132
Multi-Terminal DC Line Information .....	133
Transformer Properties .....	133
Transformer Modeling.....	133
Transformer Impedance Correction Table Display .....	135
Transformer AVR Dialog.....	135
Transformer Mvar Control Dialog.....	137
Transformer Phase Shifting Information.....	138
Transformer Fields Information Dialog .....	139
Three Winding Transformer Information.....	139
Switched Shunt Properties.....	140
Switched Shunt Information (Edit Mode).....	140
Run Mode Properties and Information .....	142
Bus Properties.....	142
Bus Information (Run Mode).....	142
Substation Properties.....	144
Substation Information (Run Mode) .....	144
Generator Properties .....	145
Generator Information .....	145
Generator Information (Run Mode) .....	145
Generator Information: MW and Voltage Control .....	146
Input-Output Curve .....	147
Load Properties .....	148
Load Information (Run Mode).....	148
Line/Transformer Properties .....	149
Line/Transformer Information Dialog (Run Mode).....	149
Transformer Impedance Correction Tables Dialog .....	152
Switched Shunt Properties.....	153
Switched Shunt Information (Run Mode).....	153
Zone Properties .....	154
Zone Information (Run Mode).....	154
General Properties and Information.....	155
Labels .....	155
Area Properties.....	156
Area Information.....	156
Area Information .....	156
Area Information: Information/Interchange .....	157
Area Information: Options .....	158
Area Information: Tie Lines .....	159
Area Information: Simulation Summary .....	159
Area Information: Scheduled MW Transactions .....	160
Area Field Information .....	160

Super Area Properties .....	162
Super Area Information Dialog.....	162
Super Area Field Information .....	163
Load Variation Properties.....	164
Load Variation Dialog .....	164
Interface Properties .....	165
Interface Information.....	165
Interface Element Information.....	167
Interface Field Information Dialog .....	168
Interface Pie Chart Information Dialog .....	168
<b>Chapter 8: Text-Based Information Displays</b>	<b>171</b>
Fundamentals.....	171
Case Information Displays .....	171
Case Information Displays .....	171
Configuring the Case Information Displays .....	171
Case Information Displays: Local Menu Options .....	173
Case Information Displays: Colors and Cell Styles.....	175
Case Information Displays: Using Cell Handles .....	175
Case Information Displays: Sorting Records .....	176
Case Information Displays: Finding Records.....	176
Filtering .....	176
Area/Zone Filters.....	176
Advanced Filtering.....	177
Advanced Filters Dialog.....	177
Advanced Filters Display .....	178
Custom Expressions.....	178
Custom Expressions.....	178
Custom Expressions Display .....	179
General .....	179
Find Dialog Basics.....	179
Model Conditions Display and Dialog.....	180
Model Filters Display and Dialog.....	180
Entering a Range of Numbers.....	181
Copying Simulator Data to and from Other Applications.....	181
Save Case Information Data.....	182
Key Fields .....	182
Customized Data Grid Display .....	183
Functions and Operators Available .....	183
Summary Displays.....	185
Case Description .....	185
Case Summary.....	186
Power Flow List .....	187
Quick Power Flow List .....	188
Making Reports .....	189
Area and Zone Displays.....	190
Area Display .....	190
Zone Display .....	191
Super Area Display.....	192
Bus Displays .....	194
Bus Display .....	194
Remotely Regulated Bus Display.....	194
Bus Mismatches Display .....	196
Substation Displays .....	196
Substation Records Display .....	196
Generator Displays .....	197

Generator Display.....	197
Generator Outages.....	198
Generator Cost Display.....	199
Load Displays.....	201
Load Display.....	201
Load Variation Schedules.....	201
Line and Transformer Displays.....	202
Line and Transformer Display.....	202
Line and Transformer Outages.....	203
Multi-Section Lines Display.....	204
Transformer Display.....	205
Three Winding Transformer Display.....	206
DC Lines Display.....	207
Multi-Terminal DC Record Display.....	207
Switched Shunt Displays.....	208
Switched Shunt Display.....	208
Interface Displays.....	209
Interface Display.....	209
Island Displays.....	210
Island Display.....	210
Transactions Displays.....	211
Base MW Transactions Display.....	211
MW Transactions Display.....	211
MW Transactions Information Dialog.....	213
Wheeling Areas Dialog.....	214
Limit Violations Displays.....	214
Limit Violations.....	214
Limit Monitoring Settings.....	216
Limit Monitoring Settings Dialog.....	217
Ownership Displays.....	219
Ownership.....	219
Owner Data Information Display.....	219
Owner Dialog.....	219
Owned Bus Records Display.....	220
Owned Load Records Display.....	220
Owned Generator Records Display.....	220
Owned Line Records Display.....	220
Update Owners Dialog.....	220
Load Flow Displays.....	220
Jacobian Display.....	220
Ybus Display.....	221
<b>Chapter 9: Injection Groups</b>	<b>223</b>
Injection Group Overview.....	223
Working with Injection Groups.....	223
Creating Injection Groups.....	223
Deleting Injection Groups.....	223
Injection Group Display.....	224
Injection Group Dialog.....	224
Injection Group Contributions By Area.....	225
Participation Groups.....	226
Participation Points Overview.....	226
Participation Point Records Display.....	226
Add Participation Points Dialog.....	227

<b>Chapter 10: Solving and Simulating a Case</b>	<b>229</b>
Options.....	229
Solution/Environment Options.....	229
Power Flow Solution Options.....	229
Power Flow Solution: General.....	230
Power Flow Solution: Automatic Controls.....	231
Power Flow Solution: Area Interchange .....	232
Power Flow Solution: Storage.....	232
Environment Options .....	232
Case Information Display Options.....	234
Template Options .....	235
Simulation Options.....	235
Limits Options.....	237
Popup Hints Options.....	237
Solution and Control .....	238
Simulation Control .....	238
Solving the Power Flow .....	240
Area Control .....	240
Set Generator Participation Factors .....	242
Transactions .....	243
Area Transaction Modeling .....	243
Area Transaction Options Dialog.....	244
Calculate MW-Distance .....	244
MW-Distance Options.....	245
Charts.....	246
Area Control Error (ACE) Chart.....	246
Area Load and Generation Chart .....	246
Area Losses Chart.....	246
Area MW Transactions Chart.....	247
Area Average Cost Chart.....	247
Bus View Online.....	247
Bus View Display.....	247
Substation View Online.....	248
Substation View Display .....	248
<b>Chapter 11: Run Mode Tools and Options</b>	<b>251</b>
General Tools .....	251
Difference Flows .....	251
Economic Curves.....	252
Movie Maker.....	253
Movie Maker Dialog.....	253
Contingency Analysis.....	255
Overview .....	255
Contingency Analysis: An Introduction .....	255
Contingency Analysis Dialog .....	255
Contingency Case Dialog .....	256
Contingency Solution Options Dialog.....	257
Contingency Element Dialog.....	257
Make-Up Power Sources.....	258
Contingency Make-Up Sources Dialog.....	259
PSS/E Contingency Format.....	259
Running the Contingency Analysis.....	259
Contingency Analysis Dialog Overview .....	260
Contingencies Tab.....	260
Contingency Definition Display .....	262

Contingency Violations Display.....	263
Lines, Buses and Interfaces Tab.....	264
Other Contingency Actions.....	265
Comparing Contingency Analysis Results.....	266
Comparing Contingencies List Displays.....	267
Options Tab.....	270
Options Tab: Modeling.....	270
Options Tab: Limit Monitoring.....	271
Options Tab: Report Writing.....	272
Options Tab: Miscellaneous.....	273
Summary Tab.....	273
Contingency Analysis Options.....	273
Auto Insert Contingencies.....	273
Loading Contingencies From a File.....	274
Saving Contingency Records to a File.....	275
Contingency Case References.....	275
Fault Analysis.....	276
Fault Analysis.....	276
Fault Analysis Dialog.....	276
Fault Analysis Bus Records.....	279
Fault Analysis Generator Records.....	279
Fault Analysis Line Records.....	279
Mutual Impedance Records.....	279
Mutual Impedance Record Dialog.....	280
Fault Analysis Load Records.....	280
Fault Analysis Switched Shunt Records.....	280
Contouring.....	280
Contouring.....	280
Contouring Options.....	281
Contour Type.....	281
Contour Type Options.....	283
Custom Color Map.....	284
Distribution Factors.....	285
Power Transfer Distribution Factors.....	285
Power Transfer Distribution Factors Dialog.....	285
Line Outage Distribution Factors (LODFs).....	288
Line Outage Distribution Factors Dialog.....	289
Advanced LODF Calculation Dialog.....	291
Directions Dialog.....	291
Directions Display.....	292
Auto Insert Directions.....	292
Sensitivities.....	293
Flow and Voltage Sensitivities.....	293
Loss Sensitivities.....	293
Transmission Loading Relief Sensitivities.....	295
TLR Sensitivities Dialog.....	295
<b>Chapter 12: PowerWorld Simulator Add-on Tools</b>	<b>297</b>
Voltage Adequacy and Stability Tool (PVQV).....	297
PowerWorld Simulator PVQV Overview.....	297
PVQV: Obtain an Initial Solution.....	298
PVQV: Consider What to Do.....	298
PVQV: Defining Injection Groups.....	298
PVQV: Defining Contingencies.....	300
PVQV: Define the Regions to Monitor.....	300
PVQV: Open the PV Study Form.....	300

PVQV: How Should the System Vary .....	301
PVQV: Options to Control the Transfer Increase .....	301
PVQV: Defining What to Monitor .....	302
PVQV: Setting Output Options .....	304
PVQV: Other Options .....	304
PVQV: Save Options for Future Use .....	305
PVQV: Performing the Analysis .....	305
PVQV: Load State, Pause, Restore Original .....	306
PVQV: The Output File .....	306
PVQV: Q-V Curves .....	306
PVQV: Q-V Curve Buses .....	306
PVQV: Q-V Curve Options .....	307
PVQV: Q-V Curve Control/Results .....	308
PVQV: Refine Model .....	308
Optimal Power Flow (OPF) .....	308
Overview .....	308
PowerWorld Simulator Optimal Power Flow Overview .....	308
OPF Objective Function .....	309
OPF Equality and Inequality Constraints .....	309
OPF Equality Constraints .....	310
OPF Inequality Constraints .....	311
Determining Set of Active Inequality Constraints .....	312
OPF Unenforceable Constraints .....	312
OPF Marginal Costs .....	313
OPF Primal LP .....	313
OPF Primal LP Single Outer Loop .....	314
OPF Future Enhancements .....	314
Options .....	315
OPF Options .....	315
OPF Options - Constraint Options .....	315
OPF Options - General Options .....	316
OPF Options - LP Basic Variables .....	318
OPF Options - LP Basis Matrix .....	318
OPF Options - Solution Results .....	318
OPF Options - Bus Marginal Price Details .....	320
OPF Records .....	320
OPF Bus Records .....	320
OPF Generator Records .....	321
OPF Line/Transformer Records .....	322
OPF Area Records .....	323
OPF Super Area Records .....	324
OPF Interface Records .....	325
Controls .....	326
OPF Controls .....	326
OPF Phase Shifter Records .....	326
OPF Restore Previous Control Settings .....	327
Examples .....	327
OPF Example - Introduction .....	327
OPF Example - Marginal Costs .....	329
OPF Example - Super Areas .....	330
OPF Example - Enforcing Line MVA Constraints .....	330
OPF Example Page 5 .....	332
Storing OPF Data .....	332
Load LP OPF Data .....	332
Store LP OPF Data .....	332
Available Transfer Capability (ATC) Analysis .....	333

Available Transfer Capability (ATC) Analysis .....	333
Available Transfer Capability Dialog.....	334
ATC Extra Monitors Dialog.....	337
Multiple Scenario Available Transfer Capability Dialog .....	338
Transfer Limiters Display .....	340
<b>Chapter 13: Auxiliary Script/Data Files</b>	<b>343</b>
Auxiliary Files.....	343
Script Section.....	344
Script Section .....	344
Script General Actions .....	345
Script General Actions.....	345
Script Edit Mode Actions.....	348
Script Edit Mode Actions.....	348
Script Run Mode Actions.....	350
Script Run Mode Actions .....	350
Script PowerFlow Submode Actions .....	353
Script Contingency Submode Actions .....	353
Script ATC Submode Actions.....	354
Script Fault Submode Actions.....	355
Script PV Submode Actions.....	356
Script QV Submode Actions .....	357
Data Section .....	358
Data Section.....	358
Data Argument List.....	358
Data Key Fields .....	360
Data List .....	360
Data SubData Sections.....	361
Data SubData Sections .....	361
Data ColorMap .....	362
Data Contingency .....	362
Data CTG Options .....	365
Data Filter .....	365
Data Gen.....	366
Data MultiSectionLine.....	367
Data InjectionGroup.....	367
Data Interface.....	368
Data ModelCondition .....	368
Data ModelFilter .....	369
Data Owner .....	369
Data DataGrid .....	370
Data ATCScenario.....	370
Data Schedule.....	371
<b>Chapter 14: PowerWorld Simulator Project Files</b>	<b>373</b>
Overview .....	373
Overview of PowerWorld Simulator Project Files.....	373
PowerWorld Project Initialization Script .....	373
Associating Project Files With Simulator .....	374
Creating a Project File.....	375
Creating a New Project File .....	375
Create Project Dialog.....	375
Opening a Project File.....	376
Opening an Existing Project.....	376

<b>Chapter 15: Web Publishing</b>	<b>377</b>
Web Publishing Overview .....	377
Saving HTML Pages in Simulator.....	377
Saving Case Information Display Contents as HTML Tables .....	377
HTML Table Format Dialog.....	377
Editing HTML Pages .....	378
HTML Editor .....	378
HTML Link Properties Dialog .....	380
HTML Image Properties Dialog.....	380
Publishing Pages to a Web Server.....	381
Publish to Web Dialog.....	381
Defining New Web Attachments.....	382
Web Publishing Status.....	383
<b>Chapter 16: Tutorials</b>	<b>385</b>
Creating a New Case .....	385
Creating a New Case Page 1 of 10 .....	385
Inserting a Bus Page 2 of 10.....	385
Inserting a Generator Page 3 of 10 .....	386
Saving the Case Page 4 of 10.....	386
Entering a Second Bus with Load Page 5 of 10.....	387
Inserting a Transmission Line Page 6 of 10.....	387
Inserting Text, Bus and Line Fields Page 7 of 10.....	389
Solving the Case Page 8 of 10.....	390
Adding a New Area Page 9 of 10 .....	391
Case Customization Page 10 of 10 .....	392
Starting with an Existing Case .....	393
Starting with an Existing Case Page 1 of 15.....	393
Loading an Existing Power Flow File Page 2 of 15.....	393
Case Information Displays Page 3 of 15.....	393
Solving the Case Page 4 of 15.....	394
Building Onelines Page 5 of 15 .....	394
Entering a Bus Page 6 of 15 .....	395
Automatic Line Insertion Page 7 of 15.....	395
Panning and Zooming Page 8 of 15.....	395
Adding Background Page 9 of 15.....	395
Simulating the Case Page 10 of 15.....	396
Run-time Object Dialogs Page 11 of 15.....	396
Online Local Menu Page 12 of 15 .....	396
Area Records Page 13 of 15.....	396
Limit Violations Page 14 of 15.....	396
Other Case Information Displays Page 15 of 15.....	397





# Chapter 1: Getting Started With PowerWorld

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

The following material is included:

- Introduction to PowerWorld Simulator
- About this Manual
- Windows Basics
- What's New in Version 8.0
- PowerWorld Simulator: Getting Started
- Using the Toolbars
- Edit and Run Mode
- Message Log
- Web Publishing
- How to Get Help

## 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 8.0 we've made Simulator easier to use, yet even more powerful and more visual.

Simulator is actually a number of integrated products. At its core is a comprehensive, robust Power Flow Solution engine capable of efficiently solving systems of up to 60,000 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 oneline diagrams with full zooming and panning capability. Moreover, system models may be modified on the fly or even built from scratch using Simulator's full-featured graphical case editor. Transmission lines may be switched in or out of service, new transmission or generation may be added, and new transactions may 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.

Simulator also provides a convenient medium for simulating the evolution of the power system over time. Load, generation, and interchange schedule variations over time may be prescribed, and the resulting changes in power system conditions may be visualized. This functionality may be useful, for example, in illustrating the many issues associated with industry restructuring.

In addition to these features, Simulator boasts integrated economic dispatch, area transaction economic analysis, power transfer distribution factor (PTDF) computation, short circuit analysis and contingency analysis, all 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.

## What's New in Version 8.0

Simulator Version 8.0 boasts a number of major new features as well as a host of smaller enhancements designed to improve the performance and convenience of the package. Among Simulator's major new features are the following:

### **Scale Injection Group Generation and Load**

Simulator can now scale injection groups of generators and loads from the Scale Case tool.

### **Substation Display**

Simulator now provides the ability to now draw oneline diagrams on a substation basis. These oneline diagrams will resemble bus oneline diagrams, only with substation objects instead of bus objects as connection points on the diagram.

### **Improved Search Functionality**

A new search tool has been added to Simulator to make find system information quicker and easier. Simulator can now search for numerous devices by name or number, and includes the ability to perform wildcard searches on lists of elements.

### **Multi-terminal DC Line Model**

Multi-terminal DC line models are now explicitly modeled in Simulator.

### **Three-winding Transformer Model**

Three-winding transformer information is now available for display in PowerWorld Simulator. The three-winding model itself is still modeled as three two-winding equivalent transformers, but now some of the original information as well as the information on the two-winding equivalents can be examined for each three-winding transformer in the case.

### **Series Capacitor Model**

Series capacitors are now distinguished in Simulator, with information on the series capacitance value and capacitor status now available for each series capacitor.

### **MW Transactions Display Grid**

A Case Information display is now available for displaying the area to area base transactions in a matrix format.

### **Ybus Display Grid**

A Case Information display is now available for displaying the load flow Ybus in a matrix format.

### **Jacobian Display Grid**

A Case Information display is now available for displaying the load flow Jacobian in a matrix format.

### **Advanced Filters Display**

A Case Information display is now available for viewing advanced filters defined for a case.

### **Model Conditions Display**

A Case Information display is now available for viewing model conditions defined for a case.

### **Model Filters Display**

A Case Information display is now available for viewing model filters defined for a case.

### **Custom Expressions Display**

A Case Information display is now available for viewing custom expressions defined for a case.

### **Customized Data Grids Display**

A Case Information display is now available for viewing customizations made for data grids of a case.

#### **Setting Custom Detail**

Online diagrams can now be customized to temporarily hide devices meeting specified criteria on a oneline diagram when in run mode.

#### **Make a Movie**

A movie making tool has been added to Simulator to allow the user to record actions and activities when solving or running simulations in run mode.

#### **Advanced LODF Calculations**

The new Advanced LODF dialog allows users to now calculate the LODF factors for a line for several different contingencies.

#### **Automatic Insertion of Switched Shunts**

Switched shunt objects can now be automatically inserted on oneline diagrams.

#### **Automatic Line Tapping**

A tool has been added to facilitate tapping a transmission line with a new bus.

#### **Automatic Bus Splitting**

A tool has been added to facilitate splitting a bus into two separate buses, including management of re-assigning connected devices to the new split devices.

## **About this Manual**

In addition to this manual, PowerWorld Simulator includes comprehensive, context-sensitive on-line help. In fact, most of this manual is derived from the on-line help. We would strongly recommend using the on-line help since it provides features not available in a printed manual, such as the ability to jump to another topic simply by double-clicking on it. Furthermore, the on-line help is likely to be the more up-to-date reference as program updates are issued.

## **Help**

On-line help is available in PowerWorld Simulator, either through using the **Help** main menu item, or by clicking the **Help** button or the **F1** key on many dialogs and displays. Furthermore, context-sensitive help is available on the oneline diagrams by positioning the mouse over any object on the oneline and pressing the **F1** key to obtain help on that object.

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

Contact our technical support at [support@powerworld.com](mailto:support@powerworld.com) for answers to your questions regarding any PowerWorld product. Or call us at (217) 265-0286.

**PowerWorld Corporation**  
2004 South Wright Street  
Suite #102  
Urbana, IL 61801

# PowerWorld Interface

## Windows Basics

Simulator 8.0 runs under Windows 95/98/2000/Me and NT 3.5 and later operating systems. 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 mouse interaction will be provided throughout this manual.

## PowerWorld Simulator: Getting Started

The key to using Simulator is to recognize that it has two distinct modes: the Edit Mode and the 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 at just about any time using the Edit Mode and Run Mode buttons on the Program Palette. Each mode has its own distinct set of menu commands. The entries on these menus are explained in the following sections.

If you are new to Simulator and seek a quick means of familiarizing yourself with it, you may wish to start with the tutorial; see *Creating a New Case*. Or if you're interested in trial-and-error learning, you may just wish to open one of the sample cases and start learning.

## Edit Mode Introduction

The Edit Mode is used to create a new case and to modify existing cases. To switch to Edit Mode, click on the *Edit Mode* button on the Program Palette, or choose **File > Switch to Edit Mode** from the file menu.

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

- Create a completely 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 *Insert Menu* for details.
- Modify the appearance of the oneline objects; see *Format Menu* 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 a more details on the Edit Mode please see *Edit Mode Overview*.

## Run Mode Introduction

The Run Mode is used to either solve a single Power Flow Solution or to run a time-domain simulation of the power system. To access the Run Mode click on the Run Mode button on the Program Palette, or choose **File > Switch to Run Mode** from the file menu.

The key menu associated with the Run Mode is the Simulation Control. This menu allows you to either perform a single Power Flow Solution (however, it is quicker to use the toolbar), or start a time-domain simulation.

Other key components of the Run Mode include:

- The oneline diagrams, which allow you to view the case graphically. See *Oneline 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.
- Various strip-charts for plotting the time-variation in system values. See *Load and Generation Chart* for an example.
- 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.
- Customization based upon whether you are doing pure power flow studies, or time-domain economic analysis.

## 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, click on **Log** on the Program Palette.

Right-clicking on the Message Log displays its local menu. To print a highlighted section of or clear the log, select either **Print Selection** or **Clear**. You can also copy the contents of the message log to the Windows clipboard by highlighting text in the log and selecting **Copy Selection to Clipboard**. Finally, you can change the Message Log's font by selecting **Change Font** from the local menu.

## Web Publishing

Simulator provides tools for creating presentations of data and diagrams produced using the application for display on the world-wide web. These tools include

- The ability to save case information display contents as HTML code.
- The ability to save oneline diagrams and strip charts as jpeg files. See *Saving Images as Jpegs* for more information.
- A basic HTML editor that can import HTML code written using other applications.
- A mechanism for uploading your HTML documents to your web server. See *Publish to Web* for details.

Consult the chapter on Web Publishing for more information on how to publish your Simulator data on-line.

# Toolbars

## Using the Toolbars

Simulator 8.0 makes extensive use of toolbars for easy access to its many features. You can move and size these toolbars according to your preferences. The toolbars house several palettes of controls, each of which may be activated with a single mouse click. Simulator provides several palettes that group commonly used functionality. Many of these toolbars are displayed by default, with certain toolbars only available in either edit mode or run mode. To either display or hide toolbars, you can either select the **Window, Toolbars** menu option, or right-click in the toolbar docking area near the top of the online diagram. The Toolbars available in Simulator are:

- Program Palette
- File Palette
- Edit Palette
- Insert Palette
- Format Palette
- Zoom Palette
- Pan/Zoom Palette
- Options/Info Palette
- Run Mode Palette

## Program Palette



The program palette gives you the ability to switch between the program's Edit and Run Modes and to control various aspects of the Power Flow Solution. The options available from the program palette include:

### **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.

### **Single Solution (Run Mode only)**

Performs a single solution of the power flow equations, as opposed to a timed simulation. The single solution button allows you to use Simulator as a standalone power flow. See PowerWorld Simulation Control for 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.

### **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.

## File Palette



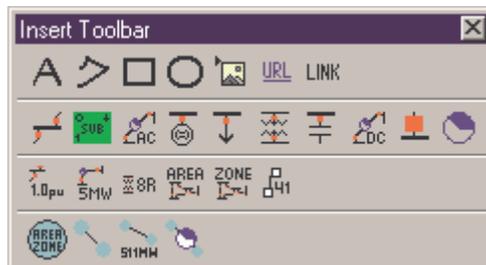
The File Palette provides access to operating system activities such as saving a oneline diagram or case model to disk, printing a oneline display to a printer, or loading a case or oneline from disk. This palette also offers access to the on-line help system and to PowerWorld's case validation tool.

## Edit Palette



The Edit Palette (Edit mode only) links to several case edit tools. You can cut or copy single objects on the oneline diagram and paste them into the same or another diagram. You can perform the same operations with groups of elements that have been identified through either the Select By Criteria or the Selection Rectangle tools.

## Insert Palette



The Insert Palette (Edit mode only) contains a number of buttons that allow you to add drawing objects to the current oneline diagram. These objects include power system components such as buses, transmission lines, transformers, loads, generators, areas, and zones, as well as informational objects such as pie charts and analogs. You may also add display objects such as text fields, rectangles, ellipses, arcs, and free-form shapes, which are not linked to objects in the power system model. The buttons on this palette provide access to most of the activities available from the Insert menu.

## Format Palette



The Format Palette (Edit mode only) allows you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. This palette also enables you to set default values for various drawing parameters and to reset the default values when necessary. The Format Palette provides access to most of the activities available from the Format branch of the main menu.

## List Display Palette



The List Display Palette provides easy access to many Case Information Display options for formatting and customizing a list display. The buttons on the toolbar will only be available if you have at least one case information display open in Simulator. The list display functions and options available from this toolbar are copying and pasting data, finding data or text, opening an information dialog, setting the display/column options, advanced filtering and sorting, refreshing the display, displaying metrics for a column, autosizing the grid, and setting the decimal places for columns containing numerical data. All of these options are available through the popup menu of each case information display as well.

## Zoom Palette



To display large detailed power systems, Simulator's onelines possess zooming and panning capabilities. The Zoom Palette 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. In addition, this toolbar enables you to save a view location, or recall a previously saved view location. This palette also links to a dialog box from which you can select a bus on which to center the display. See [Zooming and Panning](#) for more information.

## Pan/Zoom Palette



The Pan/Zoom Palette offers additional zooming and panning control. Use the four-arrow cluster to pan the display horizontally and vertically. Use the other buttons to zoom the display in and out.

## Options/Info Palette



The Options/Info Palette provides quick access to Simulator's many information displays and option settings. Use this palette to set simulation and solution options, define area/zone filters, perform a Single Power Flow Solution, generate quick power flow lists and the bus view displays, and to switch to other open oneline diagrams.

## Run Mode Palette



The Run Mode palette offers access to various Run Mode activities. It features VCR-like controls for starting, resetting, and pausing the simulation. It also links to Run Mode tools such as contouring, difference flows, and fault analysis.

## Chapter 2: Using Online Diagrams

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

The following material is included:

- Online Diagram Overview
- Online Screen Coordinates
- Online Local-Menu
- Save View Level Dialog
- Online Zooming and Panning
- Online Animation
- Print Options Dialog/Printer Setup
- Online Conditional Display of Objects
- Copying Onelines to Other Programs
- Window Menu Command
- Online Customization Options
- Relationship Between Display Objects and the Power System Model

### Online Diagram Overview

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

- zooming and panning
- conditional display of objects
- animation

Additionally, a key aspect of Simulator is the ease with which it allows you to examine and modify many of the devices contained on the online diagram. For most objects, this is accomplished by positioning the cursor on the object and right-clicking.

You may open any number of online diagrams, including multiple copies of the same oneline.

### Relationship Between Display Objects and the Power System Model

This section is intended for more advanced users familiar with the fundamentals of power system modeling.

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 display objects 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.

However, this would limit flexibility. In fact, It is quite reasonable to use more than one display object to represent a single model object. For example, by using the conditional zoom 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 the bus. Are they deleting just the Display Object or the Display Object and the Model Object? To get around this difficulty, Simulator prompts you when you are deleting a display object with an associated model object. You can either delete both the Display Object and its associated model object record, delete just the display object, or cancel the delete.

Also, 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. Also, 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 Tools and Options

### 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 to an empty portion of the oneline and then click the right mouse button. The local menu will appear. The contents of the local menu depends on whether the application is in Edit Mode or Run Mode.

#### **Print Oneline (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 Oneline** from the main menu. See [Printing Oneline 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 to a file. Images of a oneline can be saved using the bitmap format (\*.BMP), the Windows Metafile format (\*.WMF, .EMF), or the JPEG format (.jpg).

#### **Apply Template (Edit and Run Modes)**

Allows the choice of applying either a case template (see Case Templates) or a oneline template (see Oneline Templates) to the current case. Selecting this option has the same affect as selecting **Options/Tools, Apply Template** from the main menu.

**Find Bus on Oneline (Edit and Run Modes)**

Displays the Zoom, Pan and Find Options dialog. You can use this dialog to find a particular bus on the oneline.

**Pan/Zoom Control (Edit and Run Modes)**

Displays the Zoom, Pan, and Find Options dialog. This dialog allows you to modify the zooming and panning levels for the display.

**Oneline Display Options (Edit and Run Modes)**

Displays the Oneline Display Options dialog. This dialog allows you to customize the appearance of the oneline.

**Area Information Dialog (Edit and Run Modes)**

Displays the Area Information Dialog for the area of the bus that is closest to the point where the local menu was displayed.

**Contouring (Run Mode Only)**

Displays the Contour Options Dialog. This dialog allows you to contour the system voltage magnitudes or angles.

**Difference Flows (Run Mode Only)**

Displays the Difference Flows Dialog. This dialog is used to compare two power system operating points.

**Toggle Flow Visualization (Run Mode Only)**

Switches the oneline's animated flows between displaying actual flows and power transfer distribution factors.

**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.

**Set As Contingency Reference (Run Mode Only)**

This option is available only when the Contingency Analysis Dialog is open. Select this option to set the case currently in memory as the starting point for future contingency analysis.

**Close**

Closes the oneline diagram from which the local menu was invoked.

## 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 either right-clicking anywhere on an empty portion of a oneline and selecting **Oneline Display Options** from the resulting oneline local menu or selecting **Options/Tools, Oneline Display** from the main menu. This dialog houses three pages of options that govern various aspects of the oneline display. These pages are entitled Display Options, Pie Chart Options, Animated Flows Options, and Thumbnail View.

## Display Options

### Display Detail

The Display Detail box allows you to control how much information is shown on the oneline display. There are three choices:

<b>Minimal</b>	Show the oneline background, branch circuit breakers, generator MW output, and load MW/MVR.
<b>Moderate</b>	Show all Minimal information, along with bus voltages and all line flow pie charts.
<b>Complete</b>	Show all information.

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

### Set Custom Detail

Clicking this button opens the Custom Detail Dialog, which allows you to customize the oneline diagram to hide objects that do not meet specific specifications.

### Voltage Options

Voltages may be expressed in either actual units (kV) or per-unit.

### 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.

### Color Fade Value

### 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.

### Show Outaged Components with Dashed Lines

You can draw attention to branches, shunts, loads, or generators that are out of service by having Simulator draw them using dashed lines. Check this box to exploit this feature.

**Note:** Due to drawing limitations, dashed lines are always represented as a line with a thickness of 1.

### Draw an 'X' Through Off-line Generators

It may be particularly helpful for large users of Simulator to highlight pockets of generation deficiency. When this option is selected, Simulator draws a large 'X' through all off-line units so that they may be easily located on the oneline diagram.

### Enable Mouse Wheel Zooming

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

**Change in Generator Angle Per Refresh (degrees)**

Generator display objects are drawn with a dog bone rotor whose angle relative to the horizontal can be used to represent the generator internal angle. Enter a nonzero value to make use of this feature.

## Pie Chart Options

The next several fields are used to customize the appearance of the line flow pie charts.

**Style**

The Line 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, or the line amp/transformer MVA flow.

**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.

**Maximum Size**

When zooming in on a line or transformer, the size of the pie charts, like that of all other objects, increases. Eventually the pie charts will tend to dominate the display unless their size is capped. This field specifies a maximum size for the pie charts. The default value is 99.

**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 100%.

**Normal Size Scalar, Normal Color**

The standard, pre-warning fill color and scaling factor for pie charts.

**Warning Percent, Warning Size Scalar, Warning Color**

At loading percentages at or above the **Warning Percent** but below the **Limit Percent**, the size of the pie chart is automatically scaled by the amount of the **Warning Size Scalar** field. These fields permit highlighting the pie charts associated with lines and transformers that are close to their limits. The **Warning Color** changes the pie chart to this color when loading is at or above **Warning Percent**. Double-click on the color field to change the warning fill color. Default values for the Warning Percent, Warning Size Scalar, and Warning Color are 80%, 1.0, and Blue, respectively. If the **Only in Monitored Areas** option is checked, then the pie charts obey these parameters only for branches in areas for which the **Report Limit Violations** option is active. See Area Display for more information.

**Limit Percent, Limit Size Scalar, Limit Color**

At loading percentages at or above **Limit Percent**, the size of the corresponding pie chart is automatically scaled by the factor prescribed by the **Limit Size Scalar** field and the fill color becomes the color specified by the **Limit Color** field. These fields enable you to emphasize the pie charts associated with lines and transformers that are above their limits. The limit color can be changed by double-clicking on the color itself. Default values for the Limit Percent, Limit Size Scalar, and Limit Color options are 100%, 1.0, and red, respectively. If the **Only in Monitored Areas** option is checked, then the pie charts obey these parameters only for branches in areas for which the **Report Limit Violations** option is active. See Area Display for more information.

#### Only Apply 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.

#### Set color to be the same as the line...

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.

## Animated Flows Options

The next several fields are used to customize the appearance of the animated flows on the oneline.

#### Show Animated Flows

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

#### 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, and the default value is 10.

#### 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, and the default value is 10.

#### Animation Symbols

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

#### Animation Rate

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

#### Animation Parameter

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.

#### Animate Flow

Use this option to enable or disable the animated flows feature.

#### Animate Size

If checked, the size of the animation symbols will represent the quantity specified by the **Animation Parameter**. Otherwise, the symbol size will be the same for all devices regardless of the value represented.

#### Max. Zoom Level to Resize

As a oneline is zoomed, the animated flow symbols increase in size. The value of Max. Zoom Level to Resize 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.

#### Reference Values for Sizing

Maximum flow reference for sizing the animated flows. The lower MVA reference, the larger the animated flows appear on the oneline diagram.

#### Flow Visualization

Animated flows on onelines may depict either actual power flows or power transfer distribution factors, depending on what you choose here. You can also now choose to animate both the MW and MVAR flows or the MW and PTDF flows simultaneously.

#### Use Fill Color

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

#### 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 three different types of animated flows.

#### Fault Visualization Type

If the Fault Analysis tool has been used to calculate fault currents, the type (phase) of current being displayed on the oneline diagram can be changed from this box.

#### Fault P.U. Amps File Color

Shows the fill color used for the animated flow symbols if any of the fault current phase values are being displayed on the oneline diagram. Double-click on this field to change the color of the fault current flow fill color.

## 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 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 Out Multiplier

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.

#### Location

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

#### Border Width

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.

## Set Custom Detail Dialog

This dialog is opened from the Set Custom Detail button on the Online Display Options dialog. The Custom Detail Dialog allows you to choose particular regions and devices for display on a oneline diagram. The Custom Detail settings are temporary settings that allow “hiding certain parts of a oneline diagram that may not be of interest for your current needs. This option is only available during run mode.

### Area Numbers

Specify the areas for display. By default, all areas are displayed. You can assign specific areas by entering in the area numbers, separated by a dash for a range of areas, and commas for individual areas. You can also click the **Find...** button to search for areas by number or name.

### Zone Numbers

Specify the zones for display. By default, all zones are displayed. You can assign specific zones by entering in the zone numbers, separated by a dash for a range of zones, and commas for individual zones. You can also click the **Find...** button to search for zones by number or name.

### Nominal Voltage

Specify the voltage range of the devices for display. By default, all voltage levels are displayed. You can assign a specific voltage range by entering the minimum and maximum nominal voltage values.

### Type of Drawing Object

This list displays all the types of devices that can be displayed on a oneline diagram. To choose the types of devices for display, click the check box next to each type you wish to keep on the diagram. If you wish to display all the types of devices, you can click the **Check All** button or check the **Show All Types** option. If you wish to uncheck all selected types of objects, you can click the **Uncheck All** button.

When you are finished setting the Custom Detail options, you can click **OK, only show these**, and the oneline diagram will be redrawn, hiding any devices that do not meet the settings. If you wish to remove a Custom Detail filter from a oneline diagram, you can re-open the Custom Detail Dialog, and click the **Remove** button. The oneline diagram detail will be returned to normal.

## 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 **Simulation, Play**. 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 Online Display Options Dialog. To access this dialog, select **Options/Tools, Oneline Display** from the main menu, 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.
- Click on an empty portion of the oneline to display the local menu.
- Select **Copy Window 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.

## Changing the Oneline View

### 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.

### Oneline 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 Oneline Conditional Display of Objects for more information on this feature.

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

#### 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 Palette offers additional zooming options:

- Use the **Zoom in on Area** button on the *Zoom Palette* of the toolbar to select a region on which to zoom.

- Use the **Show Full Online** button on the *Zoom Palette* of the toolbar to zoom the display out to show the entire oneline.
- Use the **Pan/Zoom Control** button on the *Zoom Palette* of the toolbar to display the Pan/Zoom Dialog.
- Use the **Save View** button on the *Zoom Palette* of the toolbar to display the Save View Level Dialog, or click on the drop down arrow on the **Save View** button to choose from a list of saved views.

Finally, clicking the '+' button on the *Pan/Zoom Palette* of the PowerWorld Toolbar will zoom in on the oneline, while clicking the '-' button will zoom out.

### 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

You can also use the Pan/Zoom Palette to pan around the oneline. Simply click one of the arrow buttons to move the focal point in the desired direction.

Additionally, you can 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 **Pan/Zoom Button** on the *Zoom Palette* of the PowerWorld Toolbar or by selecting **Find Bus On Oneline** or **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 in an easily accessible list for quickly recalling defined oneline diagram views. This dialog can be called by selecting the **Save/Edit/Delete View** option from the oneline local menu, or by clicking the **Save View** button on the *Zoom Palette*.

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 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.

#### X-Coordinate

The x-coordinate for the view. This value will default to the current x-coordinate for the online 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 online 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 online 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**

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**.

## Window Menu Command

The **Window** menu tree contains several commands that affect the appearance of the windows shown on the screen, and it lists the children windows that are currently open in Simulator. To make a particular window active (and fully visible), simply click on its name.

The Window menu tree houses the following commands:

**Tile**

Rearranges the open online 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.

**Refresh Displays**

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

**Show Full**

Changes the zooming and pan levels on the currently active online diagram so that all of the objects contained on the online are visible. You could then use either the **Zoom in on Area** button on the Zoom Palette of the PowerWorld Toolbar or CTRL with the down arrow to zoom into a particular region of interest.

**Toolbars**

Provides the user control over which palettes are visible. The submenu lists all available palettes. Those that are currently visible are marked with a checkmark. Click on the name of a toolbar palette to show or to hide it.

## Online 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 from **Format, Display/Zoom Levels**.

## Printing Oneline Diagrams

### Printing Oneline Diagrams

To print a oneline diagram, select **File, Print Oneline** from the main menu, click the **Print Window** button on the File Palette, or right-click on the oneline 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 oneline 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 **File, Print Oneline** from the main menu. 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.

#### **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.



## 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:

- Opening Cases and Onelines
- Case Formats
- Starting a New Case or Oneline
- Case Validation
- Saving Cases and Onelines
- Using Templates
- Exporting Onelines in Different Graphical Formats
- Saving and Loading Auxiliary Data
- Saving Admittance Matrix and Jacobian Information

### File Menu

The file menu is used to open and save either full cases or oneline diagrams, or to create new cases. The menu is also used to validate cases or to exit Simulator. The buttons on the File Palette of the toolbar provide access to many of the same options housed by the File Menu.

### Case Formats

Simulator supports a number of case formats. You can open and save cases using any 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.

#### **PowerWorld Case (\*.pwc)**

The PowerWorld Case format is primarily included for backward compatibility with earlier versions of PowerWorld packages. While this format does not include as much information as the PowerWorld binary format, it still includes power flow data, economic parameters, and most case time variation/options values.

### PTI Raw Data Format (\*.raw) (version 26, 25, 24 and 23)

This format is included primarily for interchange of power flow data with other packages. The PTI Raw Data format only contains power flow data. The newer version 26 format, as well as the version 25, version 24 and version 23 formats are supported. When using this format, the generator participation factors are automatically set to be proportional to the MW rating for the generator.

### 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. There are two major differences between the PSLF and Simulator load flow models. First, PSLF models multi-section lines explicitly by storing a section number with transmission lines. In Simulator, the sections are converted to their own individual line record, and dummy buses are placed at each internal point of the multi-section line. Simulator also keeps track of these lines and reports them as part of a multi-section line, though each section has its own entity in the load flow data. Second, PSLF explicitly models three winding transformers. Simulator, thus far, does not explicitly model such transformers. Rather, Simulator converts the three winding transformers to an equivalent model of three (or in special cases two) two winding transformers with a common bus (also called a node.) The conversion of multi-section lines and three winding transformers is done automatically when an EPC file is opened in Simulator.

Another issue with PSLF files is that they store quite a bit of excess information that is unused in Simulator. Rather than loading all of this excess information in the computer's memory (which could hurt machine performance), the excess information is automatically stored in an extra external text file created by Simulator. This text file has an extension of .ecf, and is saved with the same name as the original case file in the same directory as the original case file. As the case is saved in Simulator, possibly with a different name, the ECF file can also have its name changed to reflect the name of the load flow file when it was saved as a PowerWorld Binary (.pwb) file. The ECF file will never be needed unless the case is written back out to PSLF EPC format, at which time Simulator will locate the ECF file, if it exists in the same directory under the same name as the PWB file, and rewrite the extra data back out to the EPC file, along with the load flow data from Simulator.

### 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.

### Auxiliary Files

The auxiliary files are used to store additional power system information that is not contained in power flow data files that do not have the pwb format. For example, generator cost information is one such set of data. Simulator uses a number of different types of auxiliary files to allow you to manipulate this data easily. These files are accessed by selecting **File, Save/Load Auxiliary Data** from the main menu to show the Save/Load Auxiliary Data display. Please note that all of the data stored in the auxiliary files is also stored in the pwb files.

## Validating a Case

Select **File, Validate Case** to check the case for errors that would prevent it from working correctly in Simulator. If a case has validation errors, you cannot switch to Run Mode. Any errors are displayed in the message log.

Case validation is performed automatically when switching from Edit Mode to Run Mode.

## Printer Setup

Choose **File, Printer Setup** from the main menu 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.

## Opening Files

### Opening a Simulation Case

The 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 **File, Open Simulation Case** from the main menu or **Open Simulation Case** button on the File Palette 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, PTI Raw Data formats version 26, version 25, version 24 and version 23, 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 should 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 Oneline Diagram

Simulator supports oneline 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, new oneline diagrams can be opened directly from any existing onelines using oneline links. However, you may also directly open a oneline diagram using the following procedure:

- Select **File, Open Oneline** from the main menu, or select the **Open Oneline** button on the File Palette.
- Choose the file type you would like to open (One Line Display File or PTI Draw File).
- In the list of available oneline files, click on the desired oneline.

- Click OK.

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

## Recently Opened Cases

A numbered list of the most recently opened cases appears in the **File Menu** immediately following the **Print** menu item. Simply click on the case's name in the menu to open the case.

# Creating Files

## Building a New Case

To create a new case in Simulator, select **File, New Case** from the main menu, or select the **New Case** button on the File Palette. 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 **File, New Oneline** from the main menu or pressing the **New Oneline** button on the File Palette. 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.

# Closing Files

## Saving Cases

Simulator allows users to save case information in several different formats. To call up the Save As dialog, select **File, Save As** from the main menu. The format in which the file will be saved depends upon the value of the **Save File Type** field shown in the lower-left corner of the dialog. Simulator can save cases in PowerWorld binary format (the default), PTI versions 23, 24, 25 or 26 raw data, 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 File, Open Simulation Case.
- In the Type of Files 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 File, Save Case As from the main menu.
- Change the Type of Files 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 **File, Save** from the main menu or the **Save Case** button on the File Palette.

## Saving a Online

Select **File, Save Online** from the main menu to save the currently selected online file. Alternatively, you may select the **Save Online** button from the File Palette.

Select **File, Save Online As** from the main menu to save the currently selected online file with a different name. You may overwrite existing files.

For both the Save Online and Save Online As options, only the online diagram is saved; the case (i.e. the power flow model) is not written to disk.

## Close Online

To close a online diagram, go to **File, Close Online**, or right-click on the background of the online diagram and select **Close** from the local menu. This will prompt you to save the online if changes have been made since the last save. Note that this does not close the current case, just the online diagram. The power system data is still loaded, and case simulations can be run with the online closed.

# Auxiliary Files

## Save/Load Auxiliary Data

The *Save/Load Auxiliary Data Display* provides access to the various supplemental data files Simulator supports. The auxiliary files are used to store additional power system information that is not contained in a conventional power flow data file, such as generator cost information and load schedules. To show this display, select **File, Save/Load Auxiliary Data**. All of the auxiliary files use a text format (i.e., they are ASCII files) so they can be easily edited. Please note that all data that can be stored in auxiliary files is also saved in the PowerWorld Binary (\*.pwb) files. The auxiliary data files are most useful when you are working with a power flow case that is not stored in a pwb file.

When saving auxiliary files, you can save the data for the entire case (the default) or you can use the bus selection page (See Bus Selection Page) to specify a desired subsystem for which to write data. When *loading* auxiliary file information, no such distinction can be made: all information contained in the file is loaded.

By default, the auxiliary files are saved in files with an extension of \*.aux. These auxiliary files contain headers that specify the type of information stored in the file. In addition, certain auxiliary data can be saved in their own file types: generator reactive capability curves with the extension \*.gcp, generator cost data with the extension \*.gcd, automatic area controls with the extension \*.aac, injection group data with the extension \*.inj, interface data with the extension \*.inf, and load schedules with the extension \*.shd. Most of these additional file types are for compatibility with auxiliary files generated in previous versions of PowerWorld Simulator.

### Identifier Format (Save Only)

When saving data, the object identifiers (such as buses) can be saved using either numbers or the object name, contained in single quotes. Likewise, when loading an auxiliary file, the identifiers can be specified with either numbers or names. Please see the specific format descriptions given below.

## Area/Zone Load Schedule Format (\*.shd)

The schedule files store the area and zone load variation schedules. The files have the following format:

For a desired area or zone, the first line of the record is either **AREA** or **ZONE**, followed by the area/zone's number or name in single quotes.

The next lines in the file must have the format *timepoint, schedule value*. The timepoint format is either hh:mm, where hh is the hour and mm is the minutes, or hh:mm:ss, where ss is the seconds. Multiple timepoints and schedule values can be on the same line.

The record is terminated by a line with single negative number.

Example

```
AREA 1
06:00 1.0000 10:00 3.0000
-1
ZONE 2
06:00 1.5000 9:00 2.5000 10:00 1.5000
-1
```

## Automatic Area Controls Format (\*.aac)

The automatic area controls file stores the control settings for areas in the case. The automatic area controls files have the following format, with one area per line:

**AREA areaname areacontroltype autotaps autoshunts**

where

<b>areaname</b>	the area's name in single quotes,
<b>areacontroltype</b>	the dispatch control for the area, in single quotes (Off AGC, Part. AGC, ED, Area Slack, OPF),
<b>autotaps</b>	the status of automatically switching taps (1 for automatically switching taps, 0 otherwise),
<b>autoshunts</b>	the status of automatically switching shunts (1 for automatically switching shunts, 0 otherwise).

## Auxiliary File Format (\*.aux)

The default auxiliary files allow for saving different types of auxiliary data in one type of data file. The auxiliary files have a format similar to an HTML file. The auxiliary files have the following format:

**<HEADER>**

**auxfiletype**

**</HEADER>**

**<DATA>**

**auxfiledata**

**</DATA>**

where

<b>auxfiletype</b>	an identifier for the type of data stored within the auxiliary file,
identifiers:	<p>AREA/ZONE LOAD SCHEDULES</p> <p>AREA AGC, AUTO TAPS, AUTO SHUNTS</p> <p>GENERATOR REACTIVE CAPABILITY CURVES</p> <p>GENERATOR COST DATA</p> <p>PARTICIPATION GROUPS</p> <p>INTERFACE RECORDS</p> <p>SEQ DATA</p>
<b>auxfiledata</b>	<p>the data for the file type, in the same formats as the format descriptions for the individual file types,</p> <p>Area/Zone Load Schedules</p> <p>Automatic Area Controls</p> <p>Generator Capability Curves</p> <p>Generator Cost Data</p> <p>Injection Groups</p> <p>Interface Data</p> <p>Sequence Data</p>

Multiple HEADER and DATA sections can be put in one auxiliary file. This allows for storing all of the data for all of the auxiliary data types in the same file. Each data type must have its own HEADER and DATA section, with the DATA section always immediately following the corresponding HEADER section.

## Auxiliary Data File Type

When you ask Simulator to open a .aux file whose type cannot be discerned, either because the header is missing or because the identifying keyword is unrecognized, Simulator will ask you to tell it what type of data it should try to import. This dialog enables you to do this. Simply select one of the four file types listed on the dialog and click ok. If the file you're trying to read doesn't conform to one of the types listed, you should click "Cancel and edit the file to ensure that it has a valid header. As of August, 2000, Simulator recognizes the following content identifiers in the header section:

- AREA AGC, AUTO TAPS, AUTO SHUNTS
- AREA / ZONE LOAD SCHEDULES
- BUS SUBSTATION RECORDS
- CROSS COMPOUND UNITS
- DIRECTION RECORDS
- AREA/ZONE FILTERS
- GENERATOR COST DATA

- GENERATOR REACTIVE CAPABILITY CURVES
- INJECTION GROUPS
- INTERFACE RECORDS
- PV CURVE OPTIONS
- SCRIPT
- SEQ DATA

## Generator Capability Curves Format (\*.gcp)

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, Q1,max,Q1,min, P2, Q2,max, Q2,min ... Pn, Qn,max, Qn,min**

where

**num/name**                    the generator's bus number or the bus' name in single quotes,  
**ID**                                the generator's single character id,  
**Pi**                                 a generator MW output value,  
**Qi,max and Qi,min**            the associated maximum and minimum reactive power limits.

Up to 10 different P/Qmax/Qmin values may be specified.

## Generator Cost Data Format (\*.gcd)

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:

**ai, bi, ci, di, FuelCost ParFac**

where

<b>ai, bi, ci, di</b>	input-output curve coefficients; see Generator Cost Information for details,
<b>fuelcost</b>	fuel cost, expressed in price per Mbtu,
<b>ParFac</b>	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 ... MWn IncCostn MWneg ParFac**

where

<b>FixedCost</b>	Operation cost independent of the generator's MW output; expressed in \$/Hr,
<b>MWi, IncCosti</b>	Output/incremental cost data point pairs that define the incremental cost model for the generator (note IncCosti has the units \$/MWhr),
<b>MWneg</b>	An arbitrary negative value used to terminate the record,
<b>ParFac</b>	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 for which it reads a cost data record onto automatic generation control (AGC), 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)

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

<b>groupname</b>	the name of the injection group, in single quotes,
<b>devtype</b>	either GEN or LOAD,
<b>busnum</b>	bus number of the device,
<b>id</b>	the device's single character ID,
<b>participation</b>	relative amount each device will contribute during a transfer,
<b>pointtype</b>	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)

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

<b>intname</b>	the interface name, in single quotes,
<b>lima, limb, limc</b>	the three limits for the interface, in MW,
<b>elemtype</b>	one of three valid element types: AREA, LINE, or ZONE
<b>from</b>	for AREA and ZONE, the first area or zone number; for LINE, the from bus number,
<b>to</b>	for AREA and ZONE, the second area or zone number; for LINE, the to bus number,
<b>id</b>	only included for LINE, the two character circuit identifier; otherwise left blank,
<b>checktoend</b>	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,
<b>fromto</b>	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

<b>busnum</b>	generator terminal bus,
<b>id</b>	generator identifier,
<b>Rpos, Xpos</b>	positive sequence resistance and reactance,

<b>Rneg, Xneg</b>	negative sequence resistance and reactance,
<b>Rzer, Xzer</b>	zero sequence resistance and reactance,
<b>RN, XN</b>	neutral-to-ground resistance and reactance.

**Load**

**LOAD busnum Gneg Bneg Gzer Bzer**

where

<b>busnum</b>	load terminal bus,
<b>Gneg, Bneg</b>	negative sequence total conductance and susceptance for all loads at bus,
<b>Gzer, Bzer</b>	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

<b>fbusnum, tbusnum</b>	from and to bus numbers for the branch,
<b>ckt</b>	branch circuit identifier,
<b>Rzer, Xzer</b>	zero sequence branch resistance and reactance,
<b>Czer</b>	total zero sequence line charging,
<b>fGzer, fBzer</b>	zero sequence line shunt conductance and susceptance at the from bus end of the branch,
<b>tGzer, tBzer</b>	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

<b>busnum</b>	terminal bus number,
<b>numblocks</b>	number of different zero sequence admittance blocks,
<b>Bzer#</b>	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

<b>from1, to1</b>	from and to bus numbers of the first mutually coupled branch,
<b>ckt1</b>	circuit identifier of the first mutually coupled branch,
<b>from2, to2</b>	from and to bus numbers of the second mutually coupled branch,
<b>ckt2</b>	circuit identifier of the second mutually coupled branch,
<b>RM, XM</b>	zero sequence mutual resistance and reactance,
<b>start1, end1</b>	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),
<b>start2, end2</b>	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 **File Menu, Export Oneline** from the main menu. 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 **File, Export Oneline** from the main menu. This brings up the **save file dialog** where you can specify file type from the dropdown menu. Choose jpeg, type the file name, and press save. This brings up the resolution screen 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 **Save As Jpeg** from the resulting local menu. Specify the compression/resolution in the Jpeg Options Dialog. Click **Cancel** to terminate the process, or click **Save** to save the jpeg to a file whose name you specify.

## 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

#### Filename for Saving Jacobian

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**.

#### Filename for Identifier Info

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.

## Templates

### Using Templates

Simulator allows you to use files called **Templates** to apply a pre-defined set of options to a particular case quickly and conveniently. Simulator supports the use of two types of templates:

- **Case Templates**, which store options related to the power flow case and the application environment. These options include schedules, case information display customizations, and other options you would typically find on the PowerWorld Simulator Options Dialog.
- **Online Display Templates**, which allow you to apply the options used for a particular online diagram to a different online diagram.

Case templates are stored as separate files having the extension *.pwt* by default. Online diagram templates are actually just PowerWorld online diagram files having the default extension *.pwd*.

For more information, see Case Templates and see Online Templates.

## Case Templates

A case template is a file of options that you can apply to the case currently in memory. You can use a case template, for example, to apply a set of options you have defined using Simulator for a previous case to a case you are just building from scratch, or to a case you have imported from another power flow program that does not support Simulator's set of options.

You can control the contents of a case template from the Templates Tab of the PowerWorld Simulator Options Dialog. A case template may contain schedules, case information display customizations, and any of the options specified on the other tabs of the PowerWorld Simulator Options Dialog.

To save a set of options as a template, select **File, Save Case As** from the main menu and change the *Save As Type* option to *PowerWorld Template (\*.pwt)*. Specify the name of the template file and click **Save**.

To apply a pre-existing case template file to the case currently in memory, select **Options/Tools, Apply Template, Case** from the main menu in either the edit or Run Modes. Select the template file whose options you wish to apply to the current case.

You can configure Simulator to apply a particular case's template options to any case you load into memory by checking the **Use Default Template** checkbox on the Template Tab of the PowerWorld Simulator Options Dialog and specifying the name of the template file. The default template file must exist in the same directory as the case to which you are applying it.

## Online Templates

A online template is really nothing more than the portion of a Simulator online diagram file that specifies its display options. To apply the display options associated with a pre-existing online diagram to the case and online currently in memory, select **Options/Tools, Apply Template, Online** from the main menu in either the run or Edit Modes. Then, select the online diagram file whose display options you wish to apply.



## 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
- Edit Mode General Procedure
- Description of the Insert Menu
- Use of Anchored Objects
- Descriptions of All Oneline Objects

## Overview

### 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 using either the PTI Raw 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 on the Program Palette, or choose **File > Switch to Edit Mode** from the file menu.

A powerful capability of the Simulator is its ability to create or modify a case by graphically placing/editing display objects on a oneline diagram. These display objects consist of both power system devices, such as buses, generators, and transmission lines; and additional objects 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**

- Select the type of the object you wish to add from the Insert Menu.
- 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 the Format Menu to change the appearance of the object once it has been placed.

#### **Moving or Resizing 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 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. Text/field objects cannot be resized; rather, you should change their fonts. You can also specify the size of most objects using their dialog boxes.

#### **Viewing/Modifying Object Parameters**

- To view and/or change the options associated with an object right-click on the object. This either displays the object's dialog box directly, or it displays the object's local menu, from which you can select 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 Font, Line/Fill, etc., by choosing **Display Appearance** from the Format Menu. To move the objects that are selected, drag them with the left mouse button down.

To select a set of objects that meet some given criteria, choose Select by Criteria from the Edit Menu. As an example, use Select By Criteria to select all the 345 kV transmission lines in a case.

You can also use the Rectangle Selector button on the Edit Palette to select all objects in a particular region of the oneline.

#### **Changing An Object's Screen Appearance**

Use the Format Menu to change the screen appearance of either a selection of objects or the entire display.

## **Insert Menu**

The Insert Menu contains the key selections for creating or modifying the oneline diagram, and hence the associated case. The options on this menu are only available in Edit Mode and after you have selected either the **New Case** command or have opened an existing case. Most of the insert activities are accessible from the Insert Palette, as well. The insert menu provides a convenient means of inserting new objects into the case by graphically placing them on the oneline diagram. See Case Information Display for details on modifying objects that are not necessarily shown on the oneline.

When adding objects to the oneline, it is helpful to consider the relationship between zoom level and display size. The objects that you add should be sized appropriately.

## 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 **Options/Tools, Oneline Display** from the main menu or choose **Oneline Display Options** from the oneline's local menu . Please see Oneline Display Options Dialog for more information.

## Anchored Objects

While in Edit Mode, Simulator allows certain objects to be attached, or *anchored*, to an 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. There are four types of anchored objects:

### Buses

Loads, generators, switched shunts, bus fields, and transmission lines may be anchored to their associated bus. When the anchor bus is moved, these anchored objects will move with it.

### Area/Zone Objects

Interface objects can be anchored to either area/zone objects or buses.

### Lines and Transformers

Circuit breakers, line pie charts, and line analogs 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 **Insert, Fields** from the main menu, 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 to ...** 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 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 noticed 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 **Options/Tools, Default Drawing Values** from the main menu and modifying the Default Drawing Options Dialog.

## Areas and Zones

### Area/Zone/Super Area Display Objects

Areas and zones may be represented on Simulator oneline diagrams. This feature may prove particularly useful for illustrating how the areas and zones comprising the system are interconnected. Not only can Simulator depict bus-to-bus connections, but it can also show area-to-area and zone-to-zone connections through the use of area and zone display objects and interface objects. Since multiple onelines can be associated with a single power flow case, you may find it useful to construct two oneline diagrams for your model: one showing the system at the bus level, and the other representing the areas or zones that make up the model as blocks tied together by area-area or zone-zone interface objects. Such oneline diagrams replace the Area Displays that were used in versions 4.1 and earlier.

#### **Edit Mode**

To add an area or zone display object to a oneline, choose **Insert, Area/Zone/Super Area** from the main menu or click the **Area/Zone** button on the Insert Palette. Then, click the left mouse on the oneline to indicate where you would like to place the new area or zone object. The Area/Zone/Super Area Display Options Dialog will appear. Supply the requested information and click *OK* to finish adding the area or zone or *Cancel* to abort the operation. If you choose *OK*, the new area or zone display object will appear on the oneline diagram at the location you chose.

When you select an area or zone display object, it will display its sizing handles. Use the sizing handles to resize the area or zone display object as desired.

Area or zone objects may be connected together by interface objects.

It is important to note that this discussion covers how to add area or zone *display objects* only, not area or zone *records*. Area or zone records cannot be added graphically. Instead, area or zone records are added to the model whenever a bus is assigned to an area or zone that does not already exist. The procedure we have discussed here can be used only to represent a pre-existing area or zone graphically on a oneline. It cannot be used to add an area or zone to a case model.

### Area/Zone/Super Area Display Options Dialog

When you try to insert an area or zone display object on a oneline diagram, Simulator opens the *Area/Zone/Super Area Display Options* dialog. Use the *Area/Zone/Super Area Display Options* dialog to control various display and identity attributes of the area or zone display object. The dialog contains the following fields:

#### **Type of Record**

Indicate whether area/zone/super area display object you are adding corresponds to an area, zone, or super area.

**Number**

This dropdown box lists the number of all areas (if you are inserting an area object), zones (if you are inserting a zone object), or super areas (if you are inserting a super area object) in the case. Use this control to associate the display object with the correct area, zone, or super area.

**Name**

If you would prefer to search through areas, zones, and super areas by name rather than by number, use the *Name* dropdown box to see a list of names of all the areas or zones in the case.

The remainder of the choices presented on the *Area/Zone/Super Area 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 **Insert, Fields, Area Fields** from the main menu, or click the **Area Fields** button on the Insert Palette. Then, click on or near 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 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.

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 **Format, Display Appearance** main menu option to change various display attributes for the area field, including its font and background color. Finally, you can use the **Format, Oneline Display, Default Font** option to change the font for all fields on the display.

## 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 modify the parameters associated with these fields. Note that the zone number itself cannot be changed on this dialog. To reach this dialog, go to **Insert, Field, Zone Field** in Edit Mode.

**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, an *Zone Number* of 0 is valid and defines the field as showing values for the entire system.

**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.

**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.

**Type of Field**

Used to determine the type of zone field to show. The two choices available are Name and Number.

**Name**

Zone name (eight characters maximum). The *Zone Number* field must correspond to a valid zone.

**Number**

Zone number (1 - 999). The *Zone Number* field must correspond to a valid area.

**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 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 losses 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, MVR 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 multiplier applied to the zone's loads. See Load Variation Display for more detail.

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 through **Insert, Fields, Super Area Field**. 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 displayed instead of a value.

# Buses

## 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 represented with either a thick horizontal line or a thick vertical line. The bus thickness and color can be customized using the Edit Mode.

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, follow this simple procedure:

- Select **Insert, Bus** from the main menu, or select the *Bus* button on the Insert Palette. 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 Box 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, and bus names can be no longer than eight characters.
- 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 a number of buses simultaneously, please see Options, Bus Renumbering Dialog.

To modify any aspect of a bus' appearance, first select the bus, and then click **Format, Display Appearance** from the main menu. You can change the length of the bus (but not its thickness) by dragging the bus' resizing handles.

To delete an exiting bus, use either the **Edit, Cut** command to preserve a copy of the bus on the Windows clipboard, or **Edit, Delete** 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 change bus device properties.

### Run Mode

For bus fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on the bus field displays the bus' local menu, from which you can elect to view the Bus Field Dialog.

### Edit Mode

Simulator 6.0 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 **Insert, Field, Bus Field** from the main menu or the **Bus Field** button from the Insert Palette 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. Specify the ID field when working with fields related to generators or loads. Next, select the type of field to show. To show load/capacitor/generator values, the corresponding device must, of course, be attached to the bus. The Setpoint Generator MW type should be selected only for cases that employ the uniform frequency option. 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 bus fields entails right-clicking the bus and selecting *Add New Fields Around Bus* from the resulting local menu. Please 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 **Format, Display Appearance** to change various display attributes of the field, including its font and background color. Select **Format, Online Display, Default Font** to change the font for all fields on a display.

## Substations

### 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.

## Generators

### 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 single 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. Furthermore, the angular position of the dog bone rotor indicates the "angle" of the voltage at the generator terminal.

Bus 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 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 **Insert, Generator** from the main menu, or click the **Generator** button on the Insert Palette. 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 single character 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 **Format, Selection** command to change the color and/or line thickness.

## Loads

### 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. You may toggle the status of the load by clicking on its associated circuit breaker.

Bus 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 the load displays the Load Dialog.

### Edit Mode

To add a new load to the case, select **Insert, Load** from the main menu, or click the **Load** button on the Insert Palette. 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 **Format, Display/Zoom Levels** from the main menu to change the drawing parameters of the load.

## Transmission Lines

### 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 Oneline 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 the **Insert, Transmission Line** command, or click the **ac Transmission Line** on the Insert Palette. 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. Choose **Format, Display/Zoom Levels** from the main menu to change the drawing parameters of the 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 dragging it with the left mouse button down. To remove a vertex, press CTRL and then click the vertex you would like to

delete. To add a vertex, press CTRL 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 MW and Mvar flow values associated with lines, transformers and dc transmission lines.

### Run Mode

Right clicking on the line field displays the Line Field Dialog.

### Edit Mode

To enter a new line field, select **Insert, Fields, Line Field** from the main menu or the **Line Fields** button on the Insert Palette 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 **Format, Display/Zoom Levels** from the main menu to change many of the line field's display attributes. Choose **Format, Online Display, Default Font** to change the font for all fields on the display.

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. Closed circuit breakers are shown as solid red squares, while open circuit breakers are shown as a green square outline. Circuit breakers cannot be placed on dc transmission lines. Clicking on the circuit breaker with the **left** mouse button changes the status of the breaker.

In PowerWorld Simulator, a circuit breaker directly controls the status of the associated line or transformer. Therefore, to open a line, you do not have to open circuit breakers at both ends of the line. In other words opening either of the line's circuit breakers opens the line. However, it is often convenient to place circuit breakers at both ends of a line.

### Edit Mode

To add a new circuit breaker to the oneline, select **Insert, Circuit Breaker** from the main menu 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 a second circuit breaker to the line, simply repeat this procedure.

To modify the parameters for an existing switch, 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 **Format, Display Appearance** from the main menu.

## 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.

Use the *Pie Chart Options* tab of the Oneline 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 **Options/Tools, Oneline Display** from the main menu or by right-clicking on the background of the oneline diagram and selecting *Oneline 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 **Insert, Line Flow Pie Chart** from the main menu or click the **Line Flow Pie Chart** button on the Insert Palette 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.

## DC Transmission Line Display Objects

DC transmission lines are represented on the onelines using multiple segment lines drawn between two buses. The line thickness and color of dc transmission lines can be customized using **Format, Display Appearance** from the main menu. 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 Oneline 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 **Insert, DC Transmission Line** from the main menu, or click the **DC Transmission Line** from the Insert Palette. 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 **Format, Display Appearance** from the main menu 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 dragging it with the left mouse button pressed. To remove a vertex, press CTRL and then click the vertex you would like to delete. To add a vertex, press CTRL and then click on the line where you would like to add a vertex.

## Transformers

### 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 Oneline 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 **Insert, Transformer** from the main menu, or click the **Transformer** button on the Insert Palette. 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  $-30^\circ$  and  $30^\circ$ ), and the step size in degrees (typical values are between  $1^\circ$  and  $2^\circ$ ).

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. Select **Format, Display Appearance** 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.

### 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 displays the Transformer Field Dialog.

### Edit Mode

To enter a new transformer field, select **Insert, Fields, Transformer Field** from the main menu or the **Transformer Field** button on the Insert Palette, 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 **Format, Display Appearance** main menu option to change various display attributes for the transformer field, including its font and background color. Finally, you can use the **Format, Online Display, Default Font** option to change the font for all fields on the display.

## Series Capacitors

### 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 Oneline 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, first select **Insert > Series Capacitor** from the main menu, or click the **Series Capacitor** button on the Insert Palette. 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 to use when the capacitor itself is **Bypassed**, and an MVA rating. On the Series Capacitor tab, enter the actual series capacitance (as a negative reactance) the series capacitor should use when the capacitor status is **In Service** (ignores r, x and c parameters).

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. Select **Format, Display Appearance** to change the series capacitor's color and/or line thickness.

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 displays the Series Capacitor Field Dialog.

### Edit Mode

To enter a new series capacitor field, select **Insert > Fields > Series Capacitor Field** from the main menu or the **Series Capacitor Field** button on the Insert Palette, 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. Use the **Format > Display Appearance** main menu option to change various display attributes for the series capacitor field, including its font and background color. Finally, you can use the **Format > Online Display > Default Font** option to change the font for all fields on the display.

## Switched Shunts

### 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.

Bus 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 bus 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 bus 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 **Insert, Switched Shunt** from the main menu, or click the **Switch Shunt** button on the Insert Palette. 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. Choose **Format, Display Appearance** from the main menu to change the color and/or line thickness.

To delete an existing switched shunt, use the **Edit, Cut** or **Edit, Delete** menu items.

## Interfaces

### 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 **Format, Display/Zoom Levels** from the main menu.

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 this interface.

#### Edit Mode

The quickest method of inserting new interface objects is to use the **Options, Auto Insert Interfaces**. Please see Automatically Inserting Interfaces for details.

To manually add a new interface object to the case, first select **Insert, Interface** from the main menu, or click the **Interface** button on the Insert Palette. 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, press CTRL and then click the vertex you would like to delete. To add a vertex, press CTRL and then click on the interface where you would like to add the vertex.

## Inserting Interfaces Dialog

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 field, go to **Case Information, Interfaces** and then right-click on the display to bring up the submenu. From the submenu choose **Auto Insert Interfaces**. This dialog is NOT brought up through **Insert, Auto Insert, Interfaces**.

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 will 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.

**Insert Interfaces**

Click this button to insert the interface records into the case.

**Cancel**

Closes the dialog without modifying the list of interfaces.

## 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 **Insert, Field, Interface Field**, 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. Use the **Format, Font** command to change the font (including its color) used with the field; use the **Make Default** and **Set to Default** to change the font for all fields on a display. Use the **Format, Line/Fill** command to change the background color used for the field.

## InterArea Flow Options Dialog

This dialog is outdated in PowerWorld Simulator version 5.0 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 **Insert, Interface Pie Chart** from the main menu or click the **Line Flow Pie Chart** button on the Insert Palette. Then, click the 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 a text file and inserts them as interface records. The format for this file should be similar to the file found at [http://www.maininc.org/pubnerc/pubnerc\\_ptdf.htm](http://www.maininc.org/pubnerc/pubnerc_ptdf.htm) (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. Flowgate contingency elements are only used during the PTDF calculations.

The Load NERC Flowgates option is available from the local menu of the Interface Records display.

## Saving NERC Flowgates

This command writes all the interfaces to a text file using the NERC flowgate format. See the file found at [http://www.maininc.org/pubnerc/pubnerc\\_ptdf.htm](http://www.maininc.org/pubnerc/pubnerc_ptdf.htm) for an example of the NERC format.

The Save NERC Flowgates option is available from the local menu of the Interface Records display.

# Oneline Links

## Links to Other Onelines

### Run Mode

Oneline 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.

Note that you can also use the **File, Open Oneline** command from the main menu to open any oneline diagram (\*.pwd) file directly. See [Opening a Oneline Diagram](#) for further details.

### Edit Mode

To add a new oneline link to a oneline diagram, first select **Insert, Oneline Link** from the main menu, or select the **Link** button on the Insert Palette. 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.

To modify the parameters of an existing oneline link, position the cursor anywhere on the object and right-click. This invokes the *Oneline Link Options Dialog*, allowing you to change the identity of the linked file and the caption. Select **Format, Display/Zoom Levels** from the main menu to modify various display attributes of the oneline link, including its font and background color. Finally, select **Format, Oneline Display, Default Font** from the main menu to modify the font for all objects on the display.

## 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 power point files or word documents or spreadsheets can also be linked.

To add a Document link to a oneline diagram, select **Insert, Document Link** from the main menu. 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 by choosing **Format, Display/Zoom Levels** from the main menu. Finally, to modify the font properties of all text objects on the oneline, including of Document objects, select **Format, Oneline Display, Default Font** from the main menu.

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.

## Background Objects and Text

### Background Lines on Onelines

The background of a oneline diagram can display added lines, polylines, and filled polygons among other items.

#### **Edit Mode**

To add a new background line, first select **Insert, Background Graphic, Background Line** from the main menu or click the **Background Line** button on the Insert Palette. To draw a series of straight line segments on the background, follow these steps:

- Position the mouse cursor where you would 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.

Use the **Format, Display/Zoom Levels** command to change the color, line thickness and fill color associated with the line. If the **Use Background Fill** option on the *Line/Fill Tab* of the Edit Multiple Objects Dialog is checked, the line is assumed to be a polygon, with a line automatically drawn between the first point on the line and the last point. The polygon is filled using the color specified in the Fill Color Field.

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 dragging it with the left mouse button down. To remove a vertex, press CTRL and then click the vertex you would like to delete. To add a vertex, press CTRL 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 Rectangles on Onelines

The background of a oneline diagram can display rectangles among other items.

### Edit Mode

To add a new rectangle, first select **Insert, Background Graphic, Background Rectangle** from the main menu. 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 it by clicking on it and then choose **Format, Display/Zoom Levels** from the main menu.

## Background Ellipses on Onelines

The background of a oneline diagram can display ellipses among other items.

### Edit Mode

To add a new ellipse, first select **Insert, Background Graphic, Background Ellipse** from the main menu, or click the **Background Ellipse** button on the Insert Palette. 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.

Select **Format, Display/Zoom Levels** from the main menu to change the color, line thickness and fill color of the ellipse.

## Background Pictures on Onelines

The background of a 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 6.0 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, select **Insert, Background Graphic, Picture** from the main menu. 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 **Format, Display/Zoom Levels** from the main menu. From the *Line/Fill 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. To change the stack level of an existing picture object, click on it to select it and then select **Format, Display/Zoom Levels** from the main menu for access to the *Line/Fill Tab*, from which you can modify the picture's stack level.

## 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 **Insert, Text** from the main menu 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.

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 **Format, Display/Zoom Levels** menu option to control the font and background color of the text object. Select **Format, Oneline Display, Default Font** from the main menu to modify the font properties for all text objects on the oneline.

## 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:

- Insertion Palettes
- Appending Additional Power Flow Components to the Existing Case
- Automatically Inserting Buses, Transmission Lines, Generators, Loads, Switched Shunts, Interfaces, Substations and Borders on the Oneline
- Renumber the Buses in a Case and/or Oneline
- Drawing Options
- Equivalencing
- Scaling the Case

### General Tools

#### 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, select **Options/Tools, Append Case** in Edit Mode.

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.

Lines and transformers are appended to a case only if both of their terminal buses exist.

#### 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, select **Options/Tools, Renumber Buses**.

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 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 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 option groups at the top of the dialog deserve further discussion.

## 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 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 Buses 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 Buses 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 Buses 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 Options/Renumber Buses from the edit-mode menu.
- Select "Load Only Buses on Oneline" and press the "Set up 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 Options/Renumber Buses from the edit-mode menu.
- Select "Freshen Current Oneline" and specify the file "oldscheme.txt" (for this example). Click the "Set up 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 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 File / Save (or File / Save As if you wish to give it a different name) from the main menu.

## 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. You will usually want to select this option.

### **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, but now have several onelines based on that case that also need to be changed.

## 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** 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** from the resulting local menu. In either case, the Bus Split 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 a 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 Simulator's main menu, 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 number 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:

- From the main menu, choose Options / Tools > Move Bus Equipment ...
- 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 Bus Split 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 Bus Split 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 most of the rest 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 an oneline diagram, the transmission line case information display, or by choosing **Tap Transmission Line** from the **Options/Tools** menu. 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. Either 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 nearest of the two buses 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 one end of the line, the **Position along line** will indicate 30% from the bus at the other end of the line, identified on the display as the **Near Bus**. If you opened the Automatic Line Tap Dialog from the case information display or the **Options/Tools** menu, 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 line shunts at the original terminal bus ends of the two new line segments. The charging capacitance of the two new elements will be set to 0.

### 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 99999. 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.

#### **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.

#### **Set bus parameters**

Click the **Tap** button to close the Automatic Line Tap Dialog 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 from Edit Mode. To open the **Default Drawing Values Dialog**, select the menu option **Options/Tools, Default Drawing Values** in Edit Mode.

The Default Drawing Values Dialog is organized using tabs showing the types of objects that can be added to onelines. Each type of object has a table which describes the defaults available for that object.

#### **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 can not 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 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

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.

#### 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. In order to specify these defaults, click on the table row corresponding to defaults you are interested in. Then click on the **Show Field Positions** button. This brings up the Positions for Adding New Fields Dialog and will show you the present settings for new fields. Positions that have a default specified will be highlights 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), the Adding New Fields Dialog will show 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 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**. 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".

Note that as you navigate through the tabs on this Default Drawing Values Dialog, the Positions for Adding New Fields Dialog will update to show you the default for the present object and voltage level you have selected.

#### 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. The specific kind of object 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.

## KV-Based Color Coding

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. The **KV-Based Color Coding Dialog** gives the user a tool for managing the colors based on kV level of the objects on the diagram.

#### kV Level

The nominal kV level.

#### Color

The background in this box represents the currently selected color for the specified kV level. To change the color, click the **Change** button and choose a different color from the color palette.

**Apply To**

The kV level color coding can be applied to five different types of oneline elements. These elements are: Lines/Transformers, Buses, Generators, Loads, and Shunts. Each object that is selected in this section will be drawn in the selected color for the specified kV level.

**Add →**

After the kV level, color, and objects to apply to are chosen, click this button to add the color coding definition to the list. You can add as many kV level specific color codes as desired. To remove a color code from the list, right-click on the color code in the list and select **Delete**.

## 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 Save/Load Auxiliary Data display to select the portion of the system to be saved in the auxiliary files ("Save) and the portion to be ignored ("Ignore).

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**

Use these lists to manually change the status of individual buses, areas, or zones. Click on the Status Field in each of these tables to toggle the object's affiliation with the study and external systems, subject to the values of the **Filter by kV** option and the **Neighbor Tiers** field.

**Set All As External**

Click this button to assign all buses to the external system.

**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.

**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.

**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 **Neighbor Tiers** controls will also shape the selection of buses to add to the external system.

**Internal**

Use these fields to specify a range of areas, zones, or buses to be added to the study system. The **Filter by kV** and **Neighbor Tiers** controls will also shape the selection of buses to add to the study system.

**Neighbor Tiers**

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.

#### 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 **Neighbor Tiers** 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, click on bus 2050 to change its status.

## List Unlinked Display Objects

Select **Options/Tools, List Display Objects, Unlinked** from the main menu in Edit Mode to identify display objects on the currently selected oneline that are not linked to a record in the power system model. 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.

The Unlinked Display Objects has the following controls:

#### 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.

#### Close, Print, Refresh, Help

Use these commands to close the dialog, to print a copy of the unlinked object list, to refresh the list, or to display this help screen.

#### Type, X/Y Location, Low/High Zoom Level

The remainder of the displays shows the type, location, and applicable zoom level for each unlinked object. This table is a type of Case Information Display and thus behaves similarly to all other case information displays. You can change the unlinked object's X and Y location coordinates, whether or not it features zoom-conditional display, and, if so, the zoom range over which it is visible. Right-click on a record in this table to invoke the local menu. Select **Show Dialog** from the local menu to view the unlinked display object's associated

information dialog, or select **Pan Online to Object** to locate and select the unlinked object on the online diagram.

## List All Display Objects

Select **Options/Tools, List Display Objects, All** from the main menu in Edit Mode to generate a list of all display objects featured on the online diagram. The **Online Objects Dialog** features a Case Information Display that identifies each display object by its type, X and Y screen coordinates, whether or not it is subject to zoom-conditional display and, if so, the zoom level range over which it is visible. All online display object attributes except *Type* can be changed from this table. Since the table is a type of Case Information Display, it exhibits behavior similar to that of all other case information displays.

Right-click on a record in the display to view the local menu. From the local menu, choose **Delete** to remove the display object from the online. Choose **Show Dialog** to show the object's corresponding information dialog. Finally, choose **Pan Online to Object** to locate and select the corresponding display object on the online.

## Insertion Using Palettes

### Palette Overview

The display object palettes are designed to help you lay out a new online 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 online, 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 **Undrawn Neighbors** list or the **All Undrawn** list to a location on the online diagram where you would like that object to appear, you can add that display object to the drawing with very little effort. The display object palettes are especially useful for adding a region of an interconnection in its entirety.

To display one of the four available palettes, select **Insert, Show Insert Palette For** from the main menu.

### Using the Insert Palettes

The display object palettes are a tool designed to accelerate the building of a online diagram based on a pre-existing power flow case. To display a palette, select **Insert, Show Insert Palette for** from the main menu.

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 online diagram. When you click on an entry in the **Displayed** list, the contents of the **Displayed Neighbors** and **Undrawn Neighbors** lists will be refreshed to identify all display 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 Display 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.

#### **Undrawn 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. 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 object 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 **Undrawn 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. 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.

Click **Close** when you are done using the display object palette.

## Automatic Object Insertion

### 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 menu and choose Insert, Auto Insert, Buses. This opens the Auto Insert Bus Dialog.

On this dialog you must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates, or if you are drawing a map of the United States, you can use 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 for the United States, 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. The buses will be drawn according to default bus object information defined in the Default Drawing Values for New Objects dialog.

Finally, if you would like Simulator to automatically insert transmission lines after completing 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.

### 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 select **Options/Tools, Auto Insert, Lines** from the main menu.

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 text fields defined in the original oneline will be displayed.

**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.

**Identifying Bus Ties**

This area deals with lines used as ties between breakers. These lines are modeled as zero impedance connections. 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 select **Options/Tools, Auto Insert, Generators** from the main menu.

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 select **Options/Tools, Auto Insert, Loads** from the main menu.

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 select **Options/Tools, Auto Insert, Switched Shunts** from the main menu.

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 generator, 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 Interfaces

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. Only area-area and zone-zone interface records can be displayed using interface display objects.

The following procedure describes the quickest way to create an interface diagram for showing PTDF information and area-area or zone-zone flows. It steps through the process from the creation of area-area or zone-zone interface records to the addition of the area or zone display objects and, finally, to the automatic insertion of interface display objects.

- Select **Case Information, Interfaces** from the main menu to bring up the Interfaces Display.
- Right-click on the Interface Records Display to show its local menu. Select **Auto Insert Interfaces** from the local menu. This generates the Auto-Insert Interfaces Dialog.
- Use the Auto-Insert Interfaces Dialog to create the area-area or zone-zone interface records automatically.

At this point, the interface *records* have been created automatically. We now want to generate an interface diagram, with the interface *display objects* added to it automatically.

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **File, New Oneline** from the main menu), place area/ zone objects at the desired locations.
- Select **Insert, Auto Insert, Interfaces** to display this dialog.
- 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.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted between any area/zone objects that have a 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 or zone-zone 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 main menu and choose **Insert, Auto Insert, Substation**. This opens the Auto Insert Substation Dialog.

On this dialog you must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates, or if you are drawing a map of the United States, you can use 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 for the United States, 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. 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 automatically insert transmission lines after completing 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.

## 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. To bring up the **Auto Insert Borders Dialog**, go to the menu and choose **Insert, Auto Insert, Borders**. This dialog has several tabs described below:

### 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.

### International

This tab allows to insert several borders which PowerWorld Corporation has drawn recently. 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](mailto:support@powerworld.com) and we'll add it to our next release.

### 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

# Equivalencing

## 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 is often desirable to reduce the size of the system model even further so that they may be solved more quickly. You can build power system equivalents in Simulator using the Equivalencing Display. The following paragraphs provides some details on this process.

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 have a circuit identifier of ‘99’. 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.

## 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 without sacrificing accuracy. For more information, please see Equivalents.

To bring up the Equivalencing Display, select **Options/Tools, Create Equivalent** from the main menu in Edit Mode. 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. 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.

#### **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.

#### **Building an Equivalent**

This procedure constructs an equivalent system. For background on equivalents, please see Equivalents. This following sections are all 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.

#### **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'.

**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.

**Delete Empty Areas and Zones**

If this option is selected, then any areas or zones that contain no buses will be removed from the case. The default value of this option is *True*. You may, however, wish to retain the empty areas or zones if they possess special attributes that you want to preserve or you plan to add buses to them after building the equivalent.

**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.

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.

## System Scaling

### 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, select **Options/Tools, Scale Case** from either Edit Mode or Run Mode. 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, Area or Zone**

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 you wish to scale load, generation, or shunt compensation at all buses in the case. If you instead wish to scale only a subset of the system, press the **Set All To No** button and begin to select the buses and areas to scale. The Power System Scaling Dialog furnishes a number a ways to do this:

- Use the **Add to be Scaled** fields to enter either a range of areas and/or buses to scale.
- Use the **Remove from Scaling** fields to enter either a range of areas and/or buses to omit from the scaling.
- Use the **Buses, Areas, Zones and Super Areas** tables to change the scaling status of individual areas or buses. Simply click on the *Scale* field for a bus or 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 changes the scaling status for all buses in the area. 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.

Finally, click **Do Scaling** to scale the load, generation, or shunt compensation.

#### **Scale by Injection Group**

If injection groups have been defined for the case, you can scale the generation and load 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 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 Generator and Load MW and Mvar 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.

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.

Finally, click **Do Scaling** to scale the load and generation for the selected injection groups.



# 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:

- Edit Menu
- Format Menu
- Changing Fonts
- Line and Fill Dialog
- Other Display Object Properties
- Background Color Dialog
- The Drawing Grid
- Zoom, Pan, and Find Dialog
- Default Drawing Values
- Inserting and Placing Multiple Display Fields

## Overview

### Edit Menu

The Edit Menu allows most of the basic editing functions. It is available only in Edit Mode. The following commands are available on the Edit Menu:

#### **Cut**

Deletes the currently selected object(s) from display. Cut objects are also copied into the paste buffer.

#### **Copy**

Copies the currently selected object(s) into the paste buffer without deleting them.

#### **Paste**

Copies the contents of the paste buffer (if any) onto the display at the current cursor location.

#### **Delete**

Deletes the currently selected object(s) from the display without also copying the object(s) to the clipboard.

#### **Select by Criteria**

Brings up the Select by Criteria Dialog which allows selection of many objects which meet a set of criteria. For example, you could select all the 345 kV transmission lines in Area X and then change them to the color blue using the Format Selection Dialog.

#### **Rectangle Selector**

The Rectangle Selector option is available only from the Edit Palette. It enables you to select all objects in a particular region of the online diagram.

## Selecting Objects

### Rectangle Selector

You can also use the Rectangle Selector button on the Edit Palette to select all objects in a particular region of the oneline. After clicking the Rectangle Selector 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 rectangle. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. Handles will appear on every object located within the selection rectangle, indicating that they have been selected.

### Select by Criteria Dialog

Selecting **Edit, Select By Criteria** from the main menu brings up the Select By Criteria Dialog. 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. 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 max and min zoom levels in the provided boxes to select only those display objects that are visible over the specified zoom range. Next, select the type of object in which you are interested from the supplied list. To select multiple items from this list, use the CTRL or SHIFT keys in tandem with the left mouse button. Display object types that are prefixed by '+', such as *Area Field (All)*, are expandable, meaning that they have several associated subtypes. If you click on a '+' sign to expand a list, you may click on the resulting '-' to compact it. If you wish to select objects that *do not* have the chosen criteria, then check **Select All Except What Meets the Above Criteria**.

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 have used the Rectangular Selector 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.

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.

# Changing Basic Display Object Properties

## Drawing Grid

The display grid is an invisible network of lines that helps you align various objects on the oneline. To enable this option, check the Snap to Grid option. The x Grid Resolution and y Grid Resolution fields determine the number of gridlines in the x and y directions.

You can temporarily disable the Snap to Grid option by holding down the ALT key while moving an object.

### Snap to Grid

If checked, then the mouse pointer will only move or “snap to intersections of the gridlines of the diagram.

### X Grid Spacing, Y Grid Spacing

The spacing between horizontal and vertical grid lines, respectively.

### Display Grid Lines on Oneline

If checked, the gridlines will become visible on the oneline diagram. The gridlines can only be visible when in Edit Mode.

### Horizontal Show Every, Vertical Show Every

Defines how many gridlines to show. Default value is 1, meaning every gridline is shown. A value of 2 shows every other grid line, and so on.

### Gridline Color

The color of the gridlines when displayed. To change the color, click in the colored edit box to open the Color dialog, and select a new color.

### Temporarily Disable Anchors

If checked, then all oneline diagram objects’ anchor property is ignored.

## Setting Background Color

The Background Color Dialog changes the background color for the display. To view this dialog, select **Format, Oneline Display, Background Color**. Click **Change Color** 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 Background Color 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 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 or area/zone object on the oneline.

#### Object Type

Select the type of object to find: *Buses, Areas, Zones, Interfaces, and Substations*.

#### Object ID

Enter the object's number or name or select the dropdown arrow to see a list of all the objects of the specified **object type** shown on the oneline.

### Find By

Used to specify whether the user is entering the object by its number or by its name. This option also determines how the entries in the **Object ID** combo box are sorted (either by number or by name).

### 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.

### Pan to Object on Oneline

Click this button to pan to the specified object. The object will be located at the center of the oneline diagram. This command does not change the zoom level.

### 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 *Oneline 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.

### Restore Default Values

Resets the zoom level and screen center to the default values.

### Set as Default Values

Sets the current zoom level and screen center as the default values for the oneline. The **Pan/Zoom to New Location** button must be clicked before this feature can be implemented.

### Pan/Zoom to New Location

Changes the screen center and zoom level to the values specified in the above fields.

## 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 **Options/Tools, Default Drawing Values** in Edit Mode.

## Format Menu

The Format Menu can be used to change the screen appearance of either individual objects or the entire display. If no items are currently selected, the only options available from the Format Menu lie in the **Oneline Display** submenu, which controls various attributes of the oneline display as whole. Otherwise, the options available from the Format Menu allow you to modify the appearance of one or more currently selected objects.

Choosing any of the first four options offered by the Format Menu brings up the Format Selection Dialog. The Format Selection Dialog is subdivided into four tabs that control different aspects of

display objects, including font, line thickness and style, fill color, zoom levels, and individual object size and display orientation.

The Format Menu offers the following options:

**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, the fill color for closed shapes, and the relative level at which an object is drawn. The Line/Fill Tab of the Format Selection Dialog is displayed.

**Display / Zoom Levels**

Simulator can hide or show display objects based on the current zoom level. Use the Zoom Levels option to designate the zoom level at which the currently selected objects are visible. The Zoom Levels Tab of the Format Selection Dialog is displayed.

**Send to Back, Bring to Front**

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.

**Oneline Display**

The Oneline Display option contains a submenu of options that control the display of the oneline diagram as a whole. These options are discussed below.

**Background Color**

Displays the Background Color Dialog. This dialog is used to change the background color of the display.

**Drawing Grid**

Displays the Drawing Grid Dialog that is used to control the drawing grid. The drawing grid is an invisible network of lines that helps you align various objects on the oneline. Use this display to change the drawing grid used for positioning objects on the display.

**Pan, Zoom and Find**

Displays the Zoom/Pan Dialog which is used to specify a desired zoom level for the display, as well as a new screen center. This dialog also allows you to pan to specific object on the oneline.

## Format Selection Dialog

The Format Selection Dialog features three tabs of controls for modifying the display attributes of selected objects.

**Font**

The Font Tab is used to change the font used in text/font objects.

#### 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.

#### Object Display

The Object Display Attribute Tab controls the size and orientation of the selected objects, as well as the range of zoom levels over which it is to be displayed.

## Font Properties

The Font Tab allows you to define the font for the selected objects. It also allows you to control the default display font through two buttons labeled **Make Default** and **Set All to Default**.

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 selecting **Format, Online Display, Default Font**.

The **Total Digits in Field** and **Digits to Right of Decimal** define the format of the number displayed by the selected field.

## 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 **Format, Line/Fill** from the main menu. 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.

#### 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, 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 Fill Color** 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.

#### Stack Level

The stack level determines how overlapping objects are shown on the display. Simulator draws Base objects first, followed by Background, Middle, and Top level objects. Thus, Base objects appear below all other objects occupying the same location, while Top level objects appear above all other objects occupying the same location. Use **Format, Send to Back** and **Format, Bring to Front** options from the main menu to adjust the display of competing objects having the same stack level.

## Other Display Object Properties

Use the Object Display Attribute Tab of the Format Selection Tab to change the display size, pixel thickness, orientation, and conditional display of the selected objects. To change the display size of the selected objects, modify the **Size** parameter. Modify the **Pixel Thickness** attribute to adjust the pixel thickness of the selected objects. To change the orientation of the selected objects, modify the **Orientation** setting. To toggle whether or not the selected objects are anchored, check or uncheck the Anchored option.

You may also use this tab to control the zoom-conditional display of selected objects. Simulator can associate with an object a zoom level range for which it will be displayed. Check the **Use Conditional Display** box to indicate that the object should be displayed only at certain zoom levels. The **Low Zoom Level** field then determines the minimum zoom level at which the object is displayed, while the **High Zoom Level** field determines the maximum zoom level at which the object is displayed.

#### Number of Selected Objects

This field shows the number of objects that are currently selected on the online.

## Cutting and Pasting

### Cut Command

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.

### Copy Command

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.

## Paste Command

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.

## Delete Command

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.

## Chapter 7: Properties of Simulator Objects

This chapter provides a description of the properties of each of the Online Display Objects. 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. 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. This chapter is divided into sections based on Edit Mode related properties, Run Mode related properties, and general properties.

### Edit Mode Properties and Information

#### Zone Properties

##### 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, first select **Case Information, Zones** to view the Zone Records Display. 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 of up to eight characters in length. You can use this field to change the zone's name, provided you click either **Save** or **OK**.

##### 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(s) to Destination Zone** button to implement the move.

##### 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.

#### Bus Properties

##### 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 99,999 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).

**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 consisting of up to eight characters.

**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.

**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 buses. **Cancel** closes the dialog without saving any changes.

**Bus Information**

**Display Shape**

Sets the appearance of the bus object.

**Pixel Thickness**

Thickness of the bus in pixels.

**Display Size**

Specifies the length of a horizontal bus, height of a vertical bus, horizontal axis of an elliptical bus, or horizontal side length of a rectangular bus.

**Display Height**

Only visible if Display Shape is Oval or Rectangle. Specifies the vertical axis of an elliptical bus or the vertical side length of a rectangular bus.

**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.

**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 999. 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.

#### Nominal Voltage

The nominal voltage for the bus in kV.

#### System Slack Bus

Check only if the bus should be modeled as a system slack bus. Each case requires at least one slack bus.

#### 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.**

#### 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 View/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

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.

## 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:

#### Bus Number

Number of the bus associated with the field. Use the dropdown box to view a list of all buses in the case with valid area/zone filters.

**Bus Name**

Name of the bus associated with the field. Use the dropdown box to view a list of all buses in the case with valid area/zone filters.

**ID**

The ID field is used when showing generator or load fields. Generator ID fields are single-character, whereas load ID's 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.

**Delta Per Mouse Click**

Bus fields can be used not only to show various fields associated with bus devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the bus field. When the up spin button is clicked, the bus field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the bus field value is decreased by the same amount.

This field is only used for fields of the following types: Load MW, Load Mvar, Setpoint Gen MW, and Switched Shunt Mvar. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the bus field on the oneline.

**Anchored**

If the *Anchored* checkbox is checked, the bus field is anchored to its associated bus, which means that it will move with the bus.

**Type of Field**

Used to determine the type of bus field to show. The following choices are available:

<b>Bus Name</b>	Name of the bus
<b>Bus Number</b>	Bus number (from 1 to 99,999)
<b>Bus Voltage</b>	Bus voltage magnitude
<b>Bus Angle</b>	Bus voltage angle in degrees
<b>Load MW</b>	Total MW load at the bus
<b>Load Mvar</b>	Total Mvar load at the bus
<b>Switched Shunt Mvar</b>	Total Mvar capacitance at the bus
<b>Gen MW Output</b>	Total MW generation at the bus
<b>Gen Mvar Output</b>	Total Mvar generation at the bus
<b>Gen AGC Status</b>	AGC status of generator; status can be toggled in Simulator
<b>Gen AVR Status</b>	AVR status of generator; status can be toggled in Simulator
<b>MW Marginal Cost</b>	Bus MW marginal cost in \$/MWh; available only with OPF
<b>Mvar Marginal Cost</b>	Bus Mvar marginal cost in \$/Mvarh; available only with OPF
<b>MW Loss Sensitivity</b>	Increase in losses for 1 MW of generation over the nominal
<b>Gen Setpoint Voltage</b>	Voltage of generator on bus
<b>Select a Field</b>	Choose from any of 42 different 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 Buses

The Shortest Path Between Buses dialog provides a way to find the shortest electrical pathway between two buses in the power system. To identify the shortest electrical path between buses, open the Bus Records display, right click on one of the bus' records you are interested in, and choose **Find Shortest Path Between Buses** from the local menu. When the Shortest Path Between Buses dialog opens, the first bus should be filled in for you. Enter the bus number of the second bus in the second field, and then click **Show Shortest Path Between Buses**. The result will be the list of the buses comprising of the shortest electrical connection between the two buses specified, in order from the starting bus to the ending bus.

## Substation Properties

### 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 **Case Information, Substations** 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.

#### Substation Buses Table

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.

**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.

## Generator Properties

### 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 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 area/zone 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 filters.

**ID**

Single-character 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.

There are four additional areas of information on this dialog for specific aspects of generation:

Display Information

Cost Data

MW Control and Voltage Control

Fault Parameters

## Generator Information: 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 Information: Cost Data

This information is located on the Generator Information Dialog.

The Input-Output Curve, Fuel-Cost, and Marginal Cost grouping 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.

### Cost Model

Simulator can model generators as 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.

### Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form  $C(P_{gi}) = (d * P_{gi}^3 + c * P_{gi}^2 + b * P_{gi} + a) * (\text{fuel cost})$ , specify the cost curve's coefficients. These coefficients can be specified only when you have chosen to use a cubic cost model.

#### Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

#### Fixed Cost

The fixed cost associated with operating the unit. This cost is independent of the generator's MW output level and is added to the cost prescribed by the piecewise linear model to obtain the total cost of operating the generator in \$/MWhr. This option can be specified only for piecewise linear cost models.

#### 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.

## Generator Information: MW Control

This information is located on the Generator Information Dialog.

The MW 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.

#### 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 PowerWorld 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.

#### 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 by using this dialog.

#### MW Ramp Limit

Specifies the maximum rate at which the real power output of the unit can be changed (in MW/minute). This rate is needed because of the mechanical and thermal stresses that arise when the output of a generator is changed. Since changing the output too quickly can damage a generator, the program will enforce this limit. You can command Simulator to ignore the limit by removing the check from the *Enforce Generator Ramp Limits* option on the Limits Tab of the PowerWorld Simulator Options Dialog. Then the output of the generator will change instantaneously.

#### 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.

#### MW Ramp Limit

When running a simulation over time, the generator cannot change by more than this value per time step. The global option for obeying generator ramp limits must be turned on.

### Generator Information: Voltage Control

This information is located on the Generator Information Dialog.

The Voltage Control grouping is used to show/change values associated with controlling the voltage/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.

#### 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 **Case Information, Others, Remotely Regulated Buses** to view the Remotely Regulated Buses Dialog, 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.

#### 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.

#### 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.

**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.

**Generator Information: 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.

**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 Cost Information**

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

$$C_i(P_{gi}) = ( a_i + b_i P_{gi} + c_i (P_{gi})^2 + d_i (P_{gi})^3 ) * \text{fuel cost} \quad \$/\text{Hour}$$

where  $P_{gi}$  is the output of the generator at bus  $i$  in MW. The values  $a_i$ ,  $b_i$ ,  $c_i$ , 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 \$/MBtu. This value varies depending on the fuel used in a generator. Typical values would be \$ 1.25/MBtu for coal and \$ 2/MBtu for natural gas. The resultant equation is known as the fuel-cost curve. The values of the  $a_i$ ,  $b_i$ ,  $c_i$ ,  $d_i$ , and the 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 (\$/MWhr) of generation, along with a fixed cost. Such curves can be defined using the Generator Information Dialog or by loading data from generator cost data files.

## Generator Participation Factors Information

The Generator Participation Factors Dialog gives you control over how the participation factors for generating units on **participation factor control** are defined. This dialog is accessed from the oneline's local submenu by choosing **Area Information Dialog** and then clicking the **Set Factors** button.

The Generator Participation Factors Dialog features the following controls:

### Set Participation Factors Based On

You can choose to assign participation factors to a group of generating units in any of four ways. The participation factor for a unit may be defined such that it is proportional to the maximum MW output of the unit. Thus, the largest units participate most heavily in implementing a scheduled change in generation. This is Simulator's default mechanism for assigning participation factors. Alternatively, you can calculate the participation factor for the unit as the difference between the maximum and current output of the unit. With this method, those units having the most reserve participate most heavily in implementing a scheduled change in generation. Furthermore, you may choose to specify a value explicitly to use as the participation factor for a group of generators by selecting the **Constant Value of** option and specifying a value in the adjacent textbox.

Finally, you may want to load participation factors for a group of units from a text file. Simply select the **File** option if this is the case, and specify the name of the file either directly in the adjacent textbox or by clicking the **Browse** button and searching for the file containing the participation factors. This file must be formatted as follows. The first line must contain either of the keywords **NUMBERS** or **NAMES** to identify how buses will be identified in subsequent lines. Then, each line must be a comma-delimited record identifying the unit's terminal bus (either by number or name, accordingly), the unit's id, and the participation factor to assign to it. No special text is used to terminate the list of participation factor records. Here is an example that assigns a participation factor of 600 to the unit 1 at both buses 18109 and 18110:

NUMBERS

18109,1,600

18110,1,600

### Set Participation Factors For

Once you have decided how to calculate the participation factors for a group of generators, you must identify the group of generators to which to apply the calculations. (Note: if you have chosen to read the factors from a file, these options are not available to you.) You have four options:

To assign participation factors to all generators in a particular area, select the **All generators in area** option and pick the desired area from the dropdown box.

To assign participation factors to all generators in a particular zone, select the **All generators in zone** option and pick the desired zone from the dropdown box.

To assign participation factors to all generators in the system regardless of area or zone affiliation and without regard to area/zone filter status, select the **All generators in system** option.

Finally, to assign participation factors only for generators with valid area/zone filters, select the **All generators with valid area/zone filters** option.

Click **Set Factors** to employ the methodology you have just outlined for assigning participation factors to a group of generating units. Select **Close** to close the dialog box after setting the factors, or **Cancel** to close the display without assigning the factors.

## 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 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 10 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 Properties

### 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, simply 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 99,999 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 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 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.

**Display Size**

Size of the load.

**Pixel Thickness**

Thickness of the display object in pixels.

**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.

**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.

**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.

**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.

## Load Modeling

Each load can be modeled as having voltage and/or time variation. The voltage variation is modeled using the Base Load Model fields on the Load Information. The time variation in the load is modeled using Area and Zone load variation schedules. Thus the actual real and reactive value of each load is determined using the following equation:

$$\text{MW} \quad \text{LoadMultiplier}(t) * (\text{SMW} + \text{IMW} * V + \text{ZMW} * V * V)$$

$$\text{Mvar} \quad \text{LoadMultiplier}(t) * (\text{SMvar} + \text{IMvar} * V + \text{ZMvar} * V * V)$$

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

The Load Multiplier field is a potentially time varying field. Its value is given by

$$\text{LoadMultiplier}(t) \quad (\text{Case Load Multiplier}) * (\text{Area Schedule Value}) * (\text{Zone Schedule Value})$$

The Case Load Multiplier allows you to scale all the loads in the case quickly and conveniently. This value is specified on the Simulation Tab of the PowerWorld Simulator Options Dialog.

The Area Schedule Value scales all of the loads in an area based upon a time-dependent schedule. These values are defined as part of a load variation schedule that can be established or modified by using the Load Schedule Dialog. Each area can have an associated load schedule, which is identified in the Area Records Display.

The Zone Schedule Value scales all of the loads in a zone based upon a time-dependent schedule. These values are defined as part of a Load Variation Schedule that can be established or modified by using the Schedule Dialog. Each zone can have an associated load schedule, which is indicated in the Zone Records display.

Note that area and zone load schedules can also be saved to and loaded from Load Schedule Files.

## Line Properties

### 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.

#### From and To Bus Nominal kV

Nominal voltage level of each terminal bus, in kV.

#### From and To Bus Area Name

Control area name in which each terminal bus is located.

#### 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.

#### Parameters / Display

#### Resistance, Reactance, Charging (B)

The resistance, reactance, and the total charging susceptance (that is,  $B$ , not  $B/2$ ) of the device (in per unit).

### **Limit A, Limit B, Limit C**

Ratings for the transmission line or transformer in MVA. Simulator allows the use of three different limits sets.

### **Status**

Current status of the device.

### **Line Shunts**

Select to view the Line Shunt Dialog. This dialog is used to change the values of the line shunts.

### **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 **Insert, Transmission Line** from the main menu).

### **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.

#### **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 Information for details on modeling a series capacitor.

#### **Fault Parameters**

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.

## 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, zig-zag (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.

## Series Capacitor Information

The Series Capacitor tab of the Line/Transformer Information dialog is only visible if the branch is a series capacitor. Series Capacitors have two additional parameters, explained below.

### 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 with a negative reactance. 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.

### Capacitor X (as negative reactance)

When the branch status is **Closed** and the capacitor status is **In Service**, the impedance used for the branch is the negative reactance (i.e. capacitance) value specified in this field. When the capacitor is in service, the resistance and shunt capacitance of the branch are considered to be 0. However, when the series capacitor is **Bypassed**, the impedance used for the bypass circuit is taken from the settings of the r, x and c fields on the Parameters tab of the Line/Transformer Information dialog. It is very important to note this functionality of a series capacitor branch. When the branch is operating as an **In Service** series capacitor, the negative reactance series capacitor value is used, and the r, x and c values on the Parameters tab are ignored. When the branch is operating as a **Bypassed** series capacitor, the r, x and c values on the Parameters tab are used, and the negative series capacitor reactance on this tab is ignored. If the branch status is **Open**, then entire circuit is treated as an open circuit, regardless of the series capacitor status.

## Multi-Section Line Information

Multi-section line records are used to group a number of 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.

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.

**Help**

Displays this help page; you may also get help by pressing the F1 key from anywhere on the dialog.

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 Information

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.

### 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 displayed field.

### Anchored

If checked, the line analog is anchored to its associated line.

### Type of Field

Used to determine the type of line field to show. The following choices are available:

<b>AC Line MW Flow</b>	MW flow into the line or transformer at the near bus
<b>AC Line Mvar Flow</b>	Mvar flow into the line or transformer at the near bus
<b>AC Line MVA Flow</b>	Magnitude of MVA flow into the line or transformer at the near bus
<b>AC Line Amp Flow</b>	Magnitude of Amp flow (in amps) into the line at the near bus
<b>AC Line MW Losses</b>	Real power losses on the line or transformer in MW
<b>AC Line Mvar Losses</b>	Reactive power losses on the line or transformer in Mvar
<b>DC Line MW Flow</b>	MW flow into the dc line at the near bus
<b>DC Line Mvar Flow</b>	Mvar flow into the dc line at the near bus
<b>MVA Limit</b>	MVA limit for the line or transformer
<b>DC Line Setpoint</b>	Set point value for the dc line; see DC Transmission Line dialog
<b>Select a Field</b>	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 Fields Information 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'.

### 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.

### Anchored

When checked, the text field will move with the series capacitor if the series capacitor is moved on the oneline diagram.

### Type of Field

Designates the type of transformer field to show. The following choices are available:

<b>Status</b>	Capacitor status
<b>Series Capacitance</b>	Capacitance of series capacitor

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

## DC Line Information

### DC Transmission Line Information

This dialog is used to view and modify the parameters associated with each dc transmission line in the system. It can also be used to insert new dc transmission lines or to delete existing ones.

This dialog has four separate pages: Line Parameters, Rectifier Parameters, Inverter Parameters, and Actual Flows. 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 Information: 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.

**Area Name**

Names of the areas in which the rectifier and inverter buses are located.

**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.

**Status**

Initial status of the dc line.

**DC Line Information: 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 Information: 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 Information: Actual Flows

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.

## Multi-Terminal DC Line Information

### 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.

**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.

## 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.

**DC Voltage**

The calculated per-unit voltage of the DC bus.

**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 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.

## Transformer Properties

### Transformer Modeling

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 (or phase shifter). 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. This type of control can be very useful in controlling the flow of real power through a transmission system.

#### 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  $-40^{\circ}$  to  $40^{\circ}$ . 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) AVR (automatic voltage regulation), 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 Active** 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 PowerWorld 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 on the Edit Mode Line/Transformer Dialog.

##### **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. For this case, clicking on the tap position with the left button raises the tap one step, while clicking on the tap position with the right button lowers the tap one step.

##### **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 regulated bus end) 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.

## 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 **Case Information > Others > Transformer Impedance Correction Tables**.

The display contains the following fields by default:

#### Table

Shows the table number for the record. Table number must be between 1 and 64. Each table record occupies two lines on the display. **1, 2, ... 11**

#### Transformer Impedance Scaling Factors

The next eleven columns show the actual fields in the table. The first line shows the off-nominal turns ratio, or phase shift angle, while the second line 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. Note that the button will not respond to the click if the *No Automatic Control* option is selected under the *Automatic Control* group.

This dialog has the following fields:

#### Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

#### Current 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.

**Minimum Voltage**

The minimum acceptable voltage at the regulated bus.

**Maximum Voltage**

The maximum acceptable voltage at the regulated bus.

**Current 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 PowerWorld 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.

### Minimum Mvar Flow, 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.

### Current 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.

### 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.

#### **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.

#### **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.

#### **Minimum MW Flow, Maximum MW Flow**

Minimum and maximum allowable MW flow through the phase shifter.

#### **Current 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 Fields Information 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'.

**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.

**Anchored**

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

**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.

### 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 Properties

### 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 99,999 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.

**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.

**Display Size**

Size of the switched shunt.

**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.

**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 PowerWorld 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.**

#### 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**. You may modify these values (except in Viewer). However, because the amount of reactive power supplied by this device changes in discrete amounts, the High Value **must be** somewhat greater than the low value. The necessary voltage range depends upon the size of the switched shunt blocks. The **Reg. Value** field shows the current per unit voltage at the regulated bus. The number of the regulated bus is shown in the **Reg. Bus #** field.

#### 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.

#### Short Circuit 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.

## Run Mode Properties and Information

### Bus Properties

#### 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 and choosing **Bus Information Dialog**. 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 99,999 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 consisting of up to eight characters. 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.

**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**

Each bus is associated with a Zone record. This field is '1' by default. 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.

**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 (degrees)**

Voltage angle at the bus in degrees. 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.

**System 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**

Click on this button to use the Quick Power Flow List to view all the flows on all devices connected to bus.

#### Device Info

##### Load Information

Displays the total MW and Mvar load at the bus. You cannot change either of these fields from this display. Select the **View/Edit Bus Load Records** to view the individual load records for the bus. Selecting this button displays the Load Dialog.

##### Generator Information

Displays the total MW and Mvar generation at the bus. You cannot change either of these fields from this display. Select the **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.

##### 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.

#### Fault Analysis Load Parameters

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 for the bus when performing an OPF solution.

## Substation Properties

### 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 **Case Information, Substations** from the main menu 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.

#### Load and Generation

Real and reactive load, generation, shunts, losses, and interchange for the substation.

#### Substation Tie Lines

The Substation Tie Line Table identifies the flows on all of the substation's ties to other substation.

## Generator Properties

### Generator Information

#### 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.

##### ID

Single-character 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.

##### 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.

##### 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.

There are two additional sections of generator information available from the Run Mode generator dialog:

MW and Voltage Control

Input-Output Curve

Short Circuit Parameters

## Generator Information: MW and Voltage Control

The **MW 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 PowerWorld 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.

### **MW Ramp Limit**

Specifies the maximum rate at which the real power output of the unit can be changed (in MW/minute). This rate is needed because of the mechanical and thermal stresses that arise when the output of a generator is changed. Since changing the output too quickly can damage a generator, the program will enforce this limit. You can command Simulator to ignore the limit by removing the check from the *Enforce Generator Ramp Limits* option on the Limits Tab of the PowerWorld Simulator Options Dialog. Then the output of the generator will change instantaneously.

### **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.

### **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, 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 **Case Information, Others, Remotely Regulated Buses** 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.

**Input-Output Curve**

The **Input-Output Curve, Fuel-Cost** and **Marginal Cost** grouping 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.

**Cost Model**

Simulator can model generators as 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

**Cubic Cost Coefficients A, B, C, D**

For cubic cost models of the form  $C(P_{gi}) = (d \cdot P_{gi}^3 + c \cdot P_{gi}^2 + b \cdot P_{gi} + a) \cdot (\text{fuel cost})$ , specify the cost curve's coefficients. These coefficients can be specified only when you have chosen to use a cubic cost model.

### Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

### Marginal Cost

Shows the marginal cost of producing real power at the generator,  $dC_i(P_{gi})/dP_{gi}$ . This field is read-only and appears 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.

### Fixed Cost

The fixed cost associated with operating the unit. This cost is independent of the generator's MW output level and is added to the cost prescribed by the piecewise linear model to obtain the total cost of operating the generator in \$/MWhr. This option can be specified only for piecewise linear cost models.

## Load Properties

### 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, simply 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 99,999 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.

#### 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...

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.

#### 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.

#### 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.

#### 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.

#### 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.

The actual load at the bus is equal to the base value multiplied by the corresponding load multiplier. The load multiplier is a potentially time varying value specifying how the load is scaled. The load multiplier depends upon the area load multiplier, the zone load multiplier, and the case load multiplier. See Load Modeling for more details. The load multiplier value cannot be changed on this dialog.

## Line/Transformer Properties

### Line/Transformer Information Dialog (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.

### Nominal kV

From and To bus nominal voltage levels.

### Area Name

Names of the areas in which the From and To buses are located.

### Parameters

#### Resistance, Reactance, Charging (B)

The resistance, reactance, and the total charging susceptance (that is, B, not B/2) of the device (in per unit).

#### Limit A, Limit B, Limit C

Ratings for the transmission line or transformer in MVA. Simulator allows the use of three different limits sets.

### Status

Current status of the device.

### Has Line Shunts

Indicates whether or not line or transformer has shunt compensation. You cannot directly change the value of this field. Rather, use the *Line Shunts / Series Capacitor* button to view/modify existing line shunts or to enter new line shunts. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer.

### Line Shunts / Series Capacitor

Select to view the Line Shunts / Series Capacitor Information Dialog. This dialog is used to change the values of the line shunts or of a series capacitor.

### Line Flow

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 PowerWorld 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.

#### **Case ... Control Enabled and Area Transformer Control Enabled**

Both of these boxes must be checked for a transformer to be controlled. **Case Control Enabled** has to be checked for any transformer in the entire case to be controlled and **Area Transformer Control Enabled** has to be checked for any transformer in that area to be controlled. Checking these does NOT automatically make the entire case or entire area on control.

#### **Automatic Control Options...**

Depending on the type of control of the transformer, this button will open up the control options for LTC Control, MVAR Control, or Phase Shifter Control.

#### **Fault Analysis Parameters**

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.

#### **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 in OPF**

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.

**Flow Limit Unenforceable**

If the line limit could not be maintained in the OPF solution, this box will be checked to indicate such. This check box cannot be changed manually.

**Line was Included in OPF Solution?**

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.

## 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 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 Properties

### 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 99,999 used to identify the bus to which the switched shunt is attached. This dropdown 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.

#### 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.

#### 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 bus' per unit voltage.

#### 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.

#### 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 PowerWorld 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.**

#### 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**. You may modify these values (except in Viewer). However, because the amount of reactive power supplied by this device changes in discrete amounts the High Value **must be** somewhat greater than the low value. The necessary voltage range depends upon the size of the switched shunt blocks. The **Reg. Value** field shows the current per unit voltage at the regulated bus. The number of the regulated bus is shown in the **Reg. Bus #** field.

#### Area Shunt Control Enabled

This checkbox provides convenient access to the *Auto Shunts* property of the shunt's corresponding area. To enforce automatic shunt control for the area, make sure that this box is checked.

#### Case Shunt Control Enabled

This checkbox provides a convenient means of disabling or enabling automatic shunt control case-wide. Leave the box unchecked to disable automatic shunt control throughout the case, regardless of the *Auto Shunt* property of the constituent areas. This control has the same effect as checking the *Disable Switched Shunt Control* checkbox on the *Power Flow Solution Tab* of the PowerWorld Simulator Options Dialog.

#### Reg. Bus P.U. Voltage to Mvar Sensitivity

The number in this area is the amount that will be added to the per unit voltage for each 1 Mvar increase by the bus controlling element.

#### Switched Shunt Blocks

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. Reactive blocks (negative shunt MVR) should be specified before capacitive blocks (positive shunt MVR).

#### Fault Analysis 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 steps and blocks.

## Zone Properties

### 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

display it from Run Mode, first select **Case Information, Zones** from the main menu to bring up the Zone Records Display. 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.

**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.

**Load Variation Model**

As with areas, the load variation within a zone is governed by a load variation schedule, which prescribes a time-dependent load multiplier to apply to the zone's load.

**Load Schedule**

This dropdown box identifies the load schedule that is currently associated with the zone. Use the dropdown box to specify the load schedule to apply to the zone from the list of defined load schedules. It is acceptable not to associate a load schedule with a zone, in which case the value specified in the **Schedule Value** field is applied at all times.

**Schedule Value**

This field shows the current load multiplier for the zone, which scales all loads in the zone. If no **Load Schedule** has been assigned to the zone, this field will display a prefix of 'N'. If the associated load schedule is disabled, the field will display a prefix of 'D'.

**Zone Tie Lines**

The Zone Tie Line Table identifies the flows on all of the zone's ties to other zones.

## General Properties and Information

### Labels

Power system equipment such as buses, generators, loads, switched shunts, areas, zones, and interfaces may have alternative names assigned to them. These alternative names are called labels. Labels allow you to refer to equipment in the power system 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.

Information dialogs corresponding to buses, generators, loads, switched shunts, 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. You can delete a label from the list by selecting it and pressing the delete key on the keyboard. You may add a label to the device by typing its name in the textbox labeled **New Label** and pressing

the **Add** button. You will not be allowed to add a Label that already exists, regardless of whether the existing label is associated with the current device or some other device. A single power system device may have multiple labels, but each label may be associated with only one device.

You also may designate a particular label to be the primary label for the device by selecting it from the list and clicking the Make Primary button. A device's primary label is the one that can be displayed individually in a Case Information Display. The other labels associated with a device can be used to import data from auxiliary data files, but they cannot be viewed individually from Case Information Displays. However, all Case Information Displays can hold a field that lists all labels assigned to a device as a semicolon-delimited string.

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 secondary labels will also provide the mapping mechanism.

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. This is the key to enabling data to be imported from an auxiliary file using labels.

## Area Properties

### Area Information

#### 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:

- 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.
- Select Case Information, Areas from the main menu. This displays the Area Display. Right-click on the record corresponding to the desired area to bring up the display's local menu, and choose Show Dialog.
- Right-click on an Area/Zone Display Object that represents an area.

The Area Information Dialog contains the following information:

#### **Number**

A dropdown box that specifies the area number. Select an area from the dropdown 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 dropdown box that specifies the area's alphabetic identifier, which may be of up to eight characters.

#### **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 dropdown 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.)

**Set 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, Simulator will assign participation factors in proportion to the maximum MW rating of each unit.

The rest of the Area Information Dialog is divided into five pages of controls:

- Information/Interchange
- Options
- Tie Lines
- Simulation Summary
- Scheduled MW Transactions

**Area Information: Information/Interchange**

The Information/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.

**Base Interchange**

Base scheduled interchange in MW between the area and all other areas (exporting power is positive). Base Interchange is interchange for which detailed time-dependent schedules are not modeled. See Area Transactions Modeling for more information on the difference between Base Interchange and Scheduled Interchange.

Use the **Base Interchange by Area Table** to set the interchange between the area and all other areas. To prescribe the 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 MW Export field for the *Unspecified* area record. To save the changes you make to this table, be sure to click **Save Changes** or **OK**. To sort the entries in any column in the Base Interchange Table, simply left- or right-click on the column title.

**Sched. Interchange (MW)**

Current scheduled interchange in MW between the area and all other areas, with power exports represented as positive. In pure power flow studies, Base Transactions are normally used instead of Scheduled Transactions, because pure power flow studies usually do not require time or cost information. For a discussion of modeling area transactions covering how

to set up scheduled transactions and the difference between this field and base interchange, see Area Transactions Modeling.

### **Area Control Error**

Current area control error (ACE) for the area in MW. Note that, when the constant frequency model is used,  $ACE = \text{area generation} - \text{area load} - \text{area losses} - \text{scheduled area interchange}$ .

### **MW 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.

### **Save Changes**

If you have made changes to the Base Interchange in the Base Interchange table, you must click the Save Changes button to save the changes to the table before displaying another area in the dialog.

## **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 zone **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 or zone 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 Schedule**

Use this dropdown box to specify the load schedule to use for the area. The load schedule prescribes how the load in the area should vary with time.

### **Load Multiplier Value**

This value is used for scaling the area load.

### **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.

### **Area Slack Bus Number**

Identifies which bus to model as the area's slack bus.

### **Generator MW Limits**

Indicates whether or not generator MW limits should be enforced for the area. If you choose *Use Case Default*, the setting specified on the *Power Flow Solution Tab* of the PowerWorld Simulator Options Dialog will be used instead.

### **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.

#### **Wheeling Charge**

The transmission price charged by the area for wheeling power between two other areas as part of a transaction.

### **Area Information: Tie Lines**

The tie line grid shows the flow on all of the tie lines for the area. The grid displays the area name, bus number, and bus name for each terminal of the tie line, with the local area's terminal listed first. The next column shows the tie line's circuit identifier. The next two columns show the real and reactive power **flowing from this area to the other area**. The next column gives the status of the tie line as open or closed. The last two columns show the real and reactive power losses. The Tie Lines display is a Case Information Display, and contains several of the options available to these types of displays.

### **Area Information: Simulation Summary**

Earlier versions of Simulator employed a distinction between *Simulation* and *Study* applications. In *Study* mode, system economics are essentially ignored. This mode was used to perform pure power flow studies for which economics were not a concern. The *Simulation* mode was commonly applied to timed simulations where area costs were a primary concern. In terms of the user interface, the only difference between the two modes involved which of the two varieties of Area Information Dialog were presented to the user. As of Simulator 6.0, the application no longer enforces this dichotomy between Study and Simulation modes. To maintain compatibility with earlier versions, however, the Simulation Summary tab has been provided to house the contents of the Simulation-mode Area Information Dialog.

#### **Total Area Cost**

Total area cost in \$ from the start of the simulation to the current time.

#### **Hourly Cost**

Current average hourly cost for the area in \$/hour. Select the **Graph Button** to the right of this field to display the Hourly Cost Chart for this area.

#### **Lambda**

Current incremental cost for the area in \$/MWH. This field is only valid if the area is on AGC.

#### **Load**

Present area load in MW. Select the **Graph Button** to the right of this field to display the Load and Generation Chart for this area.

#### **Generation**

Current generation for the area in MW. Select the Graph button to the right of this field to display the Load and Generation Chart for this area.

#### **Losses**

Current real power losses in MW for the area. Select the **Graph Button** to the right of this field to display the Area Losses Chart for this area.

#### **Scheduled Interchange**

Current scheduled interchange in MW between the area and all other areas (selling power is positive) beyond the base value. Select the **Graph Button** to the right of this field to display the Area MW Transactions Chart for this area. For a discussion of modeling area transactions, including how to set up scheduled transactions and the difference between this field and base interchange, see Area Transactions Modeling.

### Area Control Error

Current area control error (ACE) for the area in MW. When the constant frequency model is used,  $ACE = \text{area generation} - \text{area load} - \text{area losses} - \text{scheduled area interchange}$ . Select the **Graph Button** to the right of this field to display the ACE Chart for this area.

### Inadvertent

The inadvertent interchange keeps track of how much energy in MWH beyond its requirements the area has produced. The inadvertent error increases anytime the ACE is positive (indicating the area is producing too much power), while it decreases anytime the ACE is negative.

### Unserved Load

Current amount of area load that is not being served. To account for the economic costs associated with this unserved load, the area hourly cost is increased by a value equal to the amount of unserved load times the case-wide cost for unserved load (expressed in \$/MWH), which is specified on the *Simulation Tab* of the PowerWorld Simulator Options Dialog.

### Base Interchange

Base interchange in MW between the area and all other areas, with exports represented as positive. Base interchange represents real power transactions for which detailed time-dependent schedules and cost information are not modeled. See Area Transactions Modeling for more information on the difference between Study Transactions and Scheduled Transactions.

## Area Information: Scheduled MW Transactions

The Scheduled MW Transactions tab identifies the scheduled transactions in which the area participates. For more information on this table, please see MW Transactions Display.

## Area Field Information

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. Note that the area number itself cannot be changed on this dialog.

### 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.

### 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.

### 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.

**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 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.

**Type of Field**

Used to determine the type of area 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**

Total scheduled MW transactions 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 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.

Select *OK* to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

**Load Schedule Multiplier**

Indicates the current value of the multiplier applied to the zone's loads. See Load Variation Dialog for more detail.

**AGC Status**

Displays the AGC status of the area.

**Select a Field**

Choose from 40 different choices.

## Super Area Properties

### 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 the new super area.

**Delete**

Click this button to delete the super area specified in the Name field.

**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.

**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

##### 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.

##### Total ACE

The total area control error for the super area. It is calculated by summing the ACE for all areas comprising the super area.

##### Lambda

The marginal cost associated with the super area. Lambda is valid only for super areas that are on economic dispatch control.

##### Tolerance

The Area Control Error tolerance observed for the super area during a load flow solution.

##### 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.

## 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.

##### Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

##### 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.

##### 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.

## Load Variation Properties

### **Load Variation Dialog**

The Load Variation Dialog allows you to define and to modify load variation profiles. These profiles can then be assigned to specific areas or zones using either the *Options* tab of the Area Information Dialog or Zone Information Dialog or the corresponding case information display.

Load profiles, or *schedules*, are piecewise linear prescriptions of how the load multiplier of any area or zone associated with the profile should vary over time. Per the Load Modeling discussion, all loads are subject both to a *case load multiplier* and to the load multiplier of their corresponding area or zone. The Load Variation Dialog provides control over both these parameters.

The **Case Load Multiplier** entry allows you to specify the load multiplier to apply to all loads in the case. The nominal value of this parameter is 1.0. Please see Load Modeling for further details.

The next section of the dialog, entitled **Schedule**, enables you to define the load variation. The **Name** dropdown box lists all the load variations that have been defined thus far. If you are creating a new load variation, simply ignore the *Name* box; you will have the opportunity to assign a name to your new profile when you click **Save Record As**.

The **Time Scale** dropdown box controls the granularity of the schedule.

Check the *Disable* option to deactivate a currently defined schedule. Any area or zone associated with a disabled schedule will not have its load controlled by that schedule. The option to disable a schedule can be particularly useful when you want to try several different load profiles for a particular area in subsequent simulations.

The **Current Value** text box indicates the current value of the profile's prescribed load multiplier. This is a read-only field.

Simulator recognizes three types of load profiles: *daily*, *weekly*, and *weekday/weekend*. Define the load schedule as one of these classes by selecting one of the choices listed under **Schedule Type**. Daily load profiles specify how load is to vary over the course of a single day. If the simulation stretches over multiple days, the same load variation will be modeled for each subsequent day if the active load profile is of type *daily*. A *weekly* load profile is actually a composite of 7 daily load profiles. It allows you to specify load multipliers for each hour of each day of the week. If

the simulation occurs over multiple weeks, the same weekly load variation will be modeled for each subsequent week if the active load profile is of type *weekly*. Finally, a *weekday/weekend* load profile gives you the flexibility to specify different load profiles for weekdays and weekend days. If all weekdays exhibit the same load variations, specifying a weekday/weekend schedule will require less work than specifying a weekly schedule, since a weekly schedule requires you to enter a profile for each day of the week. Use a weekly profile if you want to model the variations in load peaks and shoulders exhibited by different days of the week.

Following the *Schedule Type* option is a table listing pairs of time points and load multiple values. This table defines the schedule for the load variation. If no schedule has been defined, a default schedule is shown with all values equal to 1.0. If unmodified, the default schedule is automatically deleted. If you do not wish to model a schedule, simply leave the default schedule without making any modifications.

The schedule table has a local menu that can be invoked by clicking the right mouse button. To insert a new point, choose *Insert Time Point* from the local menu, or press the **Insert Time Point** button. To delete an existing time point, right click on the record's *Load Scale* field and choose *Delete Time Point*, or press the **Delete Time Point** button. To see a plot of the load profile as it is currently defined, press the **Show Graph** button. To modify the load multiplier at a particular time, simply edit the corresponding *Load Scale* field.

Note that you can also edit the load profile from the graph that appears when you click **Show Graph**. Click on the plot of the profile to reveal its vertices. The vertices represent the time points listed in the table. When you move the mouse over a vertex, it becomes a crosshair pointer. When the mouse is in this state, you can move the vertex around the plot area to modify both the time and the load scale parameters associated with that point. You may discover and even modify the load scale associated with a vertex by right clicking the mouse when the pointer is a crosshairs. You may introduce a new time point by holding the CTRL key down and then selecting the curve with the left mouse button at the point in time where you want to define the new load scale. You can delete a vertex by holding down CTRL as you select the vertex with the left mouse key. You can adjust the scale of either axis by right-clicking on the axis. Finally, you may move the entire load profile up or down the load scale or along the time axes by selecting the curve somewhere other than at a vertex. When you are done modifying the load variation using the plot, you may close it.

To save the changes you have made to the load profile, click **Save Record**. To save the load profile to a new schedule, click **Save Record As**. Finally, to close the Load Variation Dialog without saving your changes, click **Cancel**.

## Interface Properties

### 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.

#### 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.

### Anchored

If checked, the ends of the interface will be anchored to their corresponding area, zone, or super area objects.

### Link to New Interface

Adds a new interface to the data or links a selected interface object to a new interface record.

### Interface Limit A (MW), Limit B (MW), Limit C (MW)

Specify the three possible limits for net interface flow. As for transmission lines and transformers, three distinct limits can be specified for interfaces. Which limit set is used can be controlled from the *Limits Tab* of the PowerWorld Simulator Options Dialog or the Line and Transformer Limit Violations Display.

### Direction for Flow Monitoring

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.

### MW Flow

Indicates the present net flow through the interface elements in MW. The field is valid only in Run Mode.

### 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 or the interface numbers.

### 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.

**Limit Marginal Cost**

The incremental cost of maintaining the flow on the Interface at its limit value.

**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.

**Flow Limit Unenforceable**

This field cannot be changed by the user. If the OPF solution cannot maintain the limit on the interface and reach a solved state, then this box will be checked by Simulator and the Interface limit will be violated.

**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, and inter-zone ties 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. 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. For lines and transformers, the required identifiers are **From Bus**, **To Bus**, **ID**, and at which end to monitor the flow (by default, interface branch flow is monitored at the *from end* of the branch; check the box to monitor it at the *to end*). For inter-area tie lines, specify the **From Area** and **To Area**. For inter-zone tie lines, specify the **From Zone** and **To Zone**. In the case of all three element types, when you select the *from* element, the list of available *to* elements is set automatically.

**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).

**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.

**Anchored**

If checked, the interface field is anchored to its associated interface display object.

**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, or percent loading.

## 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.

**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.

**MW Rating**

MW limit for the interface, using the current limit set for the case.

**Anchored**

If checked, the interface pie chart object is anchored to its associated interface.



## Chapter 8: Text-Based 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:

- Overview of Using the Case Information Displays
- Use of Data Filters
- Power Flow and Quick Power Flow List Displays
- Information Summary Displays
- Displays for All Major Case Components

### Fundamentals

#### Case Information Displays

Simulator offers several Case Information Displays 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. The **Case Information** menu item provides a link to these displays. Case information displays are available for buses, bus mismatches, generators, generator costs, ac lines, transformers, transformer impedance correction tables, dc lines, interfaces, areas, zones, and schedules. Other screens, such as the Area/Zone Filters and Limit Violations displays to name a couple, 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

Local Menu Options

Sorting Records

Configuring the Case Information Displays

Finding Records

#### Case Information Displays

##### 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 clicking **Display/Column Options** from the case information display's local menu.

To reset all case information display properties to their default settings, including column and data field associations, click the **Reset to Default** 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.

### Column Options

#### Available Fields Number

This list 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).

#### Show these fields in this order

This list contains the fields which are presently shown on 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 remove multiple fields (see **Remove** below) or move the order of fields (see **Move** below).

#### 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.

#### <- 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.

#### 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

#### 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.

#### 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

#### Automatically Refresh

Check this box to ensure that data presented by the case information displays always reflects the current state of the system. Generally, you will want to have the case information displays refresh automatically. However, for large systems, having several automatically refreshing case information displays open simultaneously may hamper program performance.

**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 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 Filters**

Clicking this button will open the Advanced Filters Dialog, which will allow more detailed filtering on the display.

**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 PowerWorld Simulator Options dialog box, will be reapplied.

**Row Height**

Defines the height of the rows of the case information display.

**View / Modify Custom Font**

Defines the font with which to display the records.

**Set Font / Row Height 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.

## 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. The following local menu options are common to most case information displays:

**Find**

Use the Find local menu option to retrieve a record pertaining to a particular element. Choosing Find from the local menu will open the Find Dialog Box, which may be used to find records pertaining to an element identified by either number or name.

**Quick Power Flow List**

Use the Quick Power Flow List local menu option to invoke Simulator's Quick Power Flow List tool.

**Bus View Oneline**

Use the Bus View Oneline local menu option to bring up the **Bus View Display (insert reference)**, which illustrates how the selected bus is connected to the rest of the system.

**Show Dialog**

Selecting the Show Dialog option will invoke a dialog box containing more detailed information and settings regarding the corresponding system object. For example, clicking Show Dialog from the local menu of a Bus Case Information Display will bring up the Bus Information Dialog.

### **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 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.

### **Send All to Excel**

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.

### **Send Selection to Excel**

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.

### **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. In addition, the user can choose whether to use the normal column headings or variable names 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.

### **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.

### **Print**

You can print the contents of most of the case information displays by selecting Print from the local menu.

### **Save As 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.

**Display/Column Options**

The contents and format of the information display can be controlled using the Case Information Display Dialog. See Configuring the Case Information Displays for more details.

**Advanced Filter**

Allows the user to custom filter the information in the display based on desired criteria. See Advanced Filtering for more information.

**Advanced Sort**

Allows the user to custom sort the information in the display based on desired criteria.

**Define Expression**

Allows you to define Custom Expressions that are functions of other fields.

**Refresh Display**

Select this option to update the currently displayed data to match the present state of the system.

**Close**

Closes the case information display.

**Help**

Display context-sensitive help for 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 PowerWorld Simulation Options Display, which can be invoked by selecting **Options/Tools, Solution/Environment** from the main menu.

There are three types of data fields:

<b>Standard Fields</b>	Fields that cannot be modified directly from the case information display are colored black by default.
<b>Enterable Fields</b>	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.
<b>Toggleable Fields</b>	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 PowerWorld 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.

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.

## Filtering

### Area/Zone Filters

The Area/Zone Filters Display, invoked by clicking **Case Information > Area/Zone Filters** from the main menu or the corresponding button on the Options/Info Palette, allows you to filter the information shown on the case information displays by area and zone. For small cases, you will usually not need to use this filtering capability, but it can be essential for large cases. The filters display lists each area and zone in the case, the number of buses in the area/zone, the range of bus numbers contained in the area or zone, and whether or not information about that area/zone should be displayed. In order for a device to be displayed, **both** its area and zone area/zone filter must be set.

You can switch between displaying the filters for the case areas or the zones by clicking on the associated tab.

The Area/Zone Filters screen 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 or zone by number or by name, copy records to and from other applications, and send the records to a printer. You can inspect an area or zone by selecting *Show Dialog* from the local menu, which invokes the Area Information and Zone 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 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 Filters setting for an area is *No*, then any case information display configured to enforce area/zone 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.

Click on a cell in the *Shown* field to toggle its value. Use the cell handle to propagate a particular value to multiple areas or zones, or use the *Toggle All Yes* or *Toggle All No* local menu options to set the values of all Area/Zone Filter records.

## Advanced Filtering

Simulator has always had the ability to filter data by Area and by Zone. This is known as Area/Zone Filtering. In Version 7.0 the ability to augment this filtering using Advanced Filtering was added. Advanced Filtering is accessed by 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 "(advanced filter)" in its caption if an Advanced Filter has been applied to it. To remove a filter from a Case Information Display 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 into the Advanced Filters Dialog 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

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.

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.
- 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.

Note: The comparison operation “within integer range list uses the same format as described in Entering a Range of Numbers.

Note: To add a new or delete conditions click on the **Add>>** or **Delete** button

#### 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)

#### Pre-Filter using Area/Zone Filters

When this box is checked, data is filtered first by the Area/Zone Filters and then by the Advanced Filter, therefore the data must meet both filters in order to be shown. When this box is unchecked, the Area/Zone Filters are ignored.

### Advanced Filters Display

The Advanced Filters Display is a Case Information Display available from the **Case Information > Other > Advanced Filters** menu option. 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. Advanced filters cannot be created or deleted manually from this display, but they can be saved to an auxiliary file or loaded from an auxiliary file by right-clicking on the display and choosing the appropriate option from the popup menu.

## Custom Expressions

### 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 the Case Information Display. Defining Custom Expressions is accessed by right-clicking on a Case Information Display and choosing **Define Expression** from the local menu. 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. 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. Then type in the expression as 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 you called it from 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

The Custom Expressions display can be opened by choosing the menu option **Case Information > Other > Custom Expressions**. If you have defined any custom expressions for your case, you can see the list of those expressions in this display. The name of the expression is given, along with the custom expression itself. 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 choosing to load data from an auxiliary file.

## General

### 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 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 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.

### 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 on the state of the power system.

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, use 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 you are creating a Model Condition for, 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.

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 allow Simulator to communicate data with other applications.

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 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.

**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. 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. It is also important to note that you CANNOT have columns of data that are unrecognizable by Simulator in between valid data columns in the data you are attempting to paste. Simulator will only paste columns of data from the clipboard up to the point where it finds an unrecognizable column header. If there are any valid columns of data after the unrecognized column, that data will be ignored. 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 2<sup>nd</sup> will show up.

## 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

The following table lists the key fields needed when attempting to Paste data into Case Information Displays from another data application.

Type of Data	Data Description	Required 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	
Owner	Owner Data	Owner Num		
Schedule	Schedule Records	Name		
Substations	Substation Records	Sub Num		
Swap Bus	Bus Swap Records	Present Number		
Switched Shunt	Switched Shunt Records	Number		
Transformer Correction Records	Transformer Correction Records	Table #		
Zone	Zone Records	Zone Num		

## Customized Data Grid Display

The Customized Data Grid display can be opened by choosing the menu option **Case Information > Other > Customized Data Grid**. 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.

Customized Data Grid 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 a 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 choose Save As, Auxiliary Data.

## 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	$4^5 = 1024$

*		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	$\text{min}(10,3,27,15) = 3$
MAX		Maximum value	$\text{max}(1,9)=9$ *see note
SIN		Sine	$\text{sin}(\text{pi}) = 0$ *see note
COS		Cosine	$\text{cos}(\text{pi}) = -1$
TAN		Tangent	$\text{tan}(\text{pi}) = 0$
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	$\sinh(3) = 10.01$
COSH	Hyperbolic cosine	$\cosh(2) = 3.76$
TANH	Hyperbolic tangent	$\tanh(1) = 0.76$
COTH	Hyperbolic cotangent	$\coth(1) = 1.31$
SECH	Hyperbolic secant	$\operatorname{sech}(0) = 1$
CSCH	Hyperbolic cosecant	$\operatorname{csch}(1) = 0.85$
ASINH	Hyperbolic arc sine	$\operatorname{asinh}(2) = 1.44$
ACOSH	Hyperbolic arc cosine	$\operatorname{acosh}(9) = 2.89$
ATANH	Hyperbolic arc tangent	$\operatorname{atanh}(.1) = 0.10$
ACOTH	Hyperbolic arc cotangent	$\operatorname{acoth}(7) = 0.14$
ASECH	Hyperbolic arc secant	$\operatorname{asech}(.3) = 1.87$
ACSCH	Hyperbolic arc cosecant	$\operatorname{acsch}(2) = 0.48$
ABS	Absolute value	$\operatorname{abs}(-8) = 8$
EXP	e to the power of	$\operatorname{exp}(3) = 20.08$
EXP2	2 to the power of	$\operatorname{exp2}(3) = 8$
EXP10	10 to the power of	$\operatorname{exp10}(3) = 1000$
LOG LN	Natural log	$\log(16) = 2.77$
LOG2	Log base 2	$\log_2(8) = 3$
LOG10	Log base 10	$\log_{10}(100) = 2$
CEIL	Round up	$\operatorname{ceil}(6.2) = 7$
RND	Random number	$\operatorname{rnd}(1) = .969$
INT	Truncate to an integer	$\operatorname{int}(6.8) = 6$
SGN SIGN	Sign of expression (-1, 0, or 1)	$\operatorname{sgn}(-9) = -1$
SQR SQRT	Square root	$\operatorname{sqr}(64) = 8$

## Summary Displays

### Case Description

The Case Description Dialog allows you to enter a text description of a case. The portion of the description that is actually saved with the case varies with the case type, as indicated here:

PowerWorld Binary (*.pwb)	Essentially an unlimited number of lines allowed
PTI Raw Data format (*.raw)	Two lines
GE EPC Data format (*.epc)	No case description supported

PowerWorld Case (\*.pwc) No case description supported

IEEE Common Format (\*.cf) No case description supported

In PowerWorld Viewer, these descriptions are read-only.

Select **Case Information**, **Case Description** from the main menu to display the Case Description Dialog.

## 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, select **Case Information > Case Summary** from the main menu. 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.

### Lines/Transformers

Total number of transmission lines and transformers in the case. Use the Line/Transformer Display to see a listing of these lines and transformers.

### DC Lines

Total number of dc transmission lines in the case. Use the DC Transmission Line Display to see a listing of these dc lines.

### Control Areas

Total number of control 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 PowerWorld 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 **Case Information, Interfaces** records display to open the Interface Dialog to define and modify interface objects.

### 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.

**Slack Bus Number(s)**

The number(s) of the slack bus or buses. One slack bus is required for each island. If multiple slack buses are shown, you may need to use the left/right arrow keys to view the entire field.

**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 filters.

To show this display select **Case Information, Power Flow List** from the main menu.

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 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:

- 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*.
- Select the toolbar icon from the Options/Info Palette.

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.

## Making Reports

Choosing **Make Reports** allows you to create text style reports of all power system information. This includes information regarding Areas, Zones, Buses, Generators, Loads, Switched Shunts, and Transmission Lines. These reports may be printed and customized as desired.

When you select **Case Information, Report Writer**, the **Reports Criteria Dialog** will appear. This allows you to customize the information contained in the report. By default, the Make Reports activity will create a comprehensive set of reports for all areas whose Area/Zone Filter settings allow them to be displayed. To specify which types of reports will be created and for which buses, use the options provided on the Reports Criteria Dialog.

- First, select either Area Reporting or Zone Reporting. This will designate whether to organize the report by areas or by zones.
- Next, select the areas or zones you would like to include in the report. To add or remove an area/zone from the report, click on its status with the left button to toggle between 'In Report' and 'NOT in Rep'. You may also add or remove large ranges of area/zone numbers by typing number ranges into the Add or Remove text boxes and pressing Return.
- Select the voltage levels you would like the reports to cover. The report will include records only for power system equipment whose nominal voltage falls in this range.
- Choose the **Types of Reports** you would like to create. To produce the entire set of reports (which is the default setting), select All. You can select multiple items from this list by using the shift and/or control keys.
- Select either **List Reports by Area/Zone** or **List Reports Together**. This will determine the format of the report. For instance, if you choose to List Reports Together, then all of the bus information will be grouped in one listing; whereas if you choose List Reports by Area/Zone, then the information will be grouped by Area/Zone.
- Finally, choose, **OK, Make Reports**.

A new dialog containing the report will appear. The report dialog features both a main menu and a local menu (accessible by right-clicking on the report) that allow you to change the font of the report, send it to a printer, or copy portions of the report to the Windows clipboard for use with another application.

# Area and Zone Displays

## 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 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.

To show the area records display, select **Case Information > Areas**.

By default, the area records 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 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:

<b>Off AGC</b>	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.
<b>Part AGC</b>	Area is on AGC, with generation dispatch controlled by its units' participation factors.
<b>ED</b>	Area is on economic dispatch control so that generation is dispatched in order of cost.
<b>OPF</b>	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.

### **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 base interchange for an area is set on the Area Information Dialog; the scheduled transactions for an area can be viewed using the specified MW Transactions display. See Area Transaction Modeling for a description of the difference between base and scheduled transactions.

### **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.

#### **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 PowerWorld 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 PowerWorld Simulator Options dialog must not be checked.

The local menu of the area records display has an addition 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. 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.

#### **Load Schedule**

Identifies the load variation schedule that governs how load varies in the area. The load variation schedule specifies values for the load multiplier at specific points in time. To examine the details of a load schedule or to modify its prescription, select **Case Information > Load Variation Schedules** from the main menu to bring up the Load Variation Dialog.

#### **Load Mult**

Shows the current load multiplier for the area, as constrained by the prevailing load schedule. All load in the area is scaled by this multiplier. This value of this field can be modified only through adjusting or redefining the area's associated load schedule using the Load Variation Dialog.

## 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 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 select **Case Information > Zones**.

The display contains the following fields by default:

**Zone Name**

Zone's alphanumeric identifier, eight characters maximum.

**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 MVR**

Net interchange of real and reactive power with all other zones. Exported power is assumed to be positive.

**Load Schedule**

Identifies the load variation schedule that governs how load varies in the zone. The load variation schedule specifies values for the load multiplier at specific points in time. To examine the details of a load schedule or to modify its prescription, select **Case Information > Load Variation Schedules** from the main menu to bring up the Load Variation Dialog.

**Load Mult**

Shows the current load multiplier for the zone, as constrained by the prevailing load schedule. All load in the area is scaled by this multiplier. This value of this field can be modified only through adjusting or redefining the area's associated load schedule using the Load Variation Dialog.

## 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 **Insert** menu. 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 **Case Information > Super Areas**.

The display contains the following fields by default:

**Name**

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.

**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 Displays

## 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 local menu 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 filters. Finally, when Simulator is in Edit Mode, the local menu allows you to add new buses to or remove existing buses from the system.

To show the bus records display, select **Case Information > Buses**.

By default, the bus records display contains the following fields:

**Number and Name**

Bus number between 1 and 99,999, and its alphanumeric identifier, eight characters maximum.

**Area Name**

Alphanumeric identifier of the bus' area.

**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.

**# Neighbors**

The total number of buses or "neighbors directly connected to the bus.

## Remotely Regulated Bus Display

The Remotely Regulated Bus Display provides information about all buses that are remotely regulated by one or more generators. 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 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 Online. 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 filters.

To call up the Remotely Regulated Bus Display, click **Case Information > Others > Remotely Regulated Buses**.

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.

**Gen Buses**

The next several fields list the generators that are remotely regulating this bus. Left clicking on either of these fields allows you to see all the flows into the generator’s bus using the Quick Power Flow List, while right clicking on the fields displays the Generator Dialog for the generator.

## 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 tab of the PowerWorld 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 local menu 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 Online. 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 **Case Information > Mismatches**. You can view bus mismatches only when the application is in Run Mode.

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 Displays

### 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 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 **Case Information > Substations**.

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.

**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 Displays

### 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 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 records display, select **Case Information > Generators**.

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.

**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 Outages

PowerWorld Simulator allows you to view the generators that are out of service. In order to see these generators that are currently out of service, go to **Case Information, Outages, Generators**. The Generator Outages Dialog describes the component that is out with thirteen different default properties:

**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.

**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.

Although only thirteen properties can be displayed at one time, there are over 50 different properties to choose from. To change a category, right-click on the column heading that you want to change and choose **Display/Column Options**. From there you can change the column heading by selecting a different subject from the column field drop-down menu.

The outages can be organized according to any column by simply clicking on the title of the column. The first click will order them in increasing order if the information is numeric and alphabetical order if the information is words. The second click on a column will order the rows in reverse order from the first click.

## Generator Cost Display

The Generator Fuel Cost Display presents more detailed cost information for each generator in the case. The Generator Fuel 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 Records Display according to the constraints of the area/zone 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 cost records display, select **Case Information > Generator Fuel Costs**.

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**

Name of the area containing the generator.

**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.

**IOA, IOB, IOC, IOD, FuelCost**

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 type.

**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.

**Lambda**

Incremental cost multiplied by the penalty factor. The penalty factor is defined as being  $1/(1 + \partial P_{loss}/\partial P_{gi})$ . Like the **LossSens** field, this field is only calculated when the generator's area is on economic dispatch control. When operating at the economic dispatch solution, all of the generators in an area will have equal lambda values (in the absence of congestion).

# Load Displays

## 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 local menu from which you can print, copy, and modify its information as well as view the information dialog of its associated loads. The local menu 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 filters.

To show the load display, select **Case Information > Loads**.

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 and time 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 Variation Schedules

The Load Variation Schedules Display identifies load variation profiles that have been defined. These profiles can be assigned to areas or zones to prescribe how the load within the areas or zones should vary over time. PowerWorld employs a load model in which all the loads in the case are first scaled from their Base Case values by the Case Load Multiplier. This provides a convenient means of changing the load level for the entire case. Furthermore, for each area and zone the loads can be modeled to change proportionally with time (i.e., they are all scaled from their base values by the same value) according to a piecewise linear model whose values can be displayed and modified using the Schedules Dialog. The Load Variation Display identifies

information that specify various piecewise linear variations that can be applied to the loads in an area or zone.

The Load Variation Schedules 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 load profiles. The local menu also affords the opportunity to define new load profiles or to delete existing ones, or to enable or disable a particular profile. You can also sort the load variation information 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 Display according to the constraints of the area/zone filters.

To show the load display, select **Case Information > Others > Load Variation Schedules**. The load display contains the following fields by default:

**Name**

The name of the load profile schedule. This name may contain up to 24 characters.

**Enabled**

Indicates whether the current load variation will be applied to any area or zone that has been defined to use it. An area may be associated with a particular load variation through either the *Options Tab* of the Area Information Dialog or the *Load Schedule* field of the Area Case Information Display. Likewise, zones may be associated with a particular load variation through either the *Options Tab* of the Zone Information Dialog or the *Load Schedule* field of the Zone Case Information Display.

**Value**

The current value of the load multiplier prescribed by the schedule.

**Type**

Simulator recognizes three types of load profiles. Daily load profiles describe a load variation that is common to all days for which the simulation is performed. Weekly load profiles chart load variation across an entire week. Finally, weekend load profiles permit a distinction between how load is to vary during a weekday and how it is to vary on a weekend day. The *Type* field will thus contain one of three values: *daily*, *weekly*, or *weekend*.

**# Time Points**

Indicates the number of segments in the piecewise linear model. A load profile having  $n$  time points defines the load multiplier at  $n$  points in time. Time points may be added or deleted using the Load Schedule Dialog.

## Line and Transformer Displays

### 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 filters.

To show the line/transformer display, select **Case Information > Lines and Transformers**.

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.

**From MW, From Mvar, From MVA (Run Mode default)**

Real, reactive, and complex power flowing into the line at the *from bus*.

**Lim MVA, Max Percent**

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 (Run Mode 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.

**Resistance, Reactance, Capacitance (Edit Mode default)**

Indicates the branch's R, X, and line charging in ohms.

**Lim A MVA, Lim B MVA, Lim C MVA (Edit Mode default)**

Identifies the limit set for the line or transformer. All limits are expressed in MVA.

See Transformer Display for a listing of transformer specific fields.

## Line and Transformer Outages

PowerWorld Simulator allows you to view the lines and transformers that are out of service. In order to see these transformers and lines, go to **Case Information, Outages, Line/Transformers**. The Line and Transformer Outages Dialog describes the component that is down with six properties.

Although only six properties can be displayed at one time, there are over 80 different properties to choose from. To change a category, right-click on the column heading that you want to change

and choose **Display/Column Options**. From there you can change the column heading by selecting a different subject from the column field drop-down menu.

The outages can be organized according to any column by simply clicking on the title of the column. The first click will order them in increasing order if the information is numeric and alphabetical order if the information is words. The second click on a column will order the rows in reverse order from the first click.

## Multi-Section Lines Display

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 filters.

To show this display select **Case Information > Others > Multi-Section Lines**.

The display contains the following fields by default:

**From Bus #, From Name**

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*.

**To Bus #, To Name**

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.

**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 Oneline. 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 **Case Information > Transformers**.

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

<b>Fixed</b>	The tap positions are fixed
<b>LTC</b>	The tap ratio changes to regulate bus voltage
<b>Phase</b>	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 PowerWorld 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**

Error between the regulated value and 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, select **Case Information > Others > 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.

**Pri, Sec and Ter MW and MVA**

These fields display the real and reactive power delivered at each of the three winding transformer terminals.

## 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 filters.

To show the dc line records display, select **Case Information > DC Lines**.

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 Number, Rect 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 Oonline from the local menu to view additional information about the rectifier bus.

### Inv Number, Inv 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 Oonline 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

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.

### 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 **Case Information > Other > Multi-terminal DC Lines** from the main menu bar.

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.

**Converter Bus**

The number of the AC converter bus at which the DC voltage is controlled.

## Switched Shunt Displays

### 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 Online. 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 filters.

To call up the Switched Shunt Display, click **Case Information > Switched Shunts**.

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. Only one switched shunt is allowed per bus. Use the right menu options to inspect either the Quick Power Flow List or the Bus View Display for the terminal bus.

**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 PowerWorld Simulator Options Dialog. This Control Mode field is toggleable.

**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.

**Max Mvar, Min Mvar**

The maximum and minimum Mvar range for the switched shunt.

## Interface Displays

### 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 and/or transformers, 2) total tie-lines between two adjacent areas, and 3) total tie-lines between two adjacent zones. 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. Interface information can also be extremely useful for summarizing the flows occurring on a large network. Interface flows can be monitored using the Limit Violations 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 filters.

To show the interface display, select **Case Information > Interfaces**.

Interface information can be saved in the “\*.inf auxiliary file format. See Save/Load Auxiliary Data for details.

The Interface display contains the following fields:

**Number**

Numeric identifier for the interface.

**Name**

Alphanumeric identifier for the interface (24 characters maximum).

**MW Flow**

Current MW flow on the interface.

**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, To - From, or Both.

**MW A Limit, MW B Limit, MW C Limit**

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.

## Island Displays

### 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 **Case Information > Islands**.

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.

## Transactions Displays

### Base MW Transactions Display

The Base MW Transactions Display is a quick way to view the defined base transactions between areas within the load flow case. The display is setup 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 base 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 base transactions from the case.

### MW Transactions Display

The MW Transactions Display lists the MW transactions that have been defined for the case. Please note that this display shows only the *scheduled* transactions between areas, not the base transactions. See Area Transaction Modeling for a description of the difference between the two.

The MW 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, and modify its information as well as view the MW Transaction Information dialog of its associated transactions. The local menu also affords the opportunity to define new MW transactions using the same MW Transaction Information Dialog or to delete existing ones. You can also sort the MW transaction 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 filters.

To call up the MW Transactions Display, click **Case Information > MW Transactions** from Run Mode.

This MW Transactions Display contains the following fields by default:

**Seller Area Name, Buyer Area Name**

Names of the selling and buying areas.

**Type**

Designates the firmness of the transaction: either firm (non-recallable) or nonfirm (recallable).

**Enabled**

Indicates whether the transaction is active or inactive. Inactive transactions will not be modeled as part of the simulation. This is a toggleable field. For inactive transactions, the information fields are printed in gray text.

**Sched MW**

Scheduled MW value for the transaction. The scheduled MW will equal the actual MW for the transaction except during ramp periods. The duration of the ramp periods can be specified from the Simulation tab of the PowerWorld Simulator Options Dialog. You can also disable the enforcement of generator ramp limits from the Limits tab of the same dialog.

**Actual MW**

Actual MW value for the transaction. When a transaction is inactive, its actual MW value is zero.

**Buy Price (\$/MWhr)**

Price the buyer pays for the transaction in \$/MWhr. The seller receives this amount, minus any wheeling charges.

**Start Time, End Time**

Starting and ending time for the transaction. To model transactions without a starting or ending time, model it as a base transaction.

**Day Type**

Designates how to model the transaction for different days of the week. This field can assume one of four possible values. A value of *Specified Days* forces Simulator to model the transaction only for the days specified in the **Start Day** and **End Day** fields. A value of *Week Days* indicates that the transaction will be modeled every day Monday through Friday, but not on the weekends. Transactions with a Day Type of *Every Day* are modeled both on weekdays and weekend days. Finally, transactions with a Day Type of *Weekend* occur only on the weekend. This is a toggleable field.

**Start Day, End Day**

For transactions having a Day Type of *Specified Days*, the Start Day and End Day fields define the days for which the transaction will be modeled. These fields are ignored if the transaction's Day Type is not *Specified Days*.

**Wheeling Cost**

The amount of the buy price being paid as Wheeling charges.

## MW Transactions Information Dialog

The MW Transactions Information Dialog is used to modify or insert new scheduled transactions. You can display this dialog using the MW Transactions list display. The dialog has the following fields:

### Buyer Area, Seller Area

Shows the buying and selling area associated with the transaction. You can **only modify** these fields when you are inserting a new transaction from the MW Transactions list display. Otherwise to change either the buyer or the seller you need to first remove the transaction and then insert a new one.

### Transaction Amount (MW)

MW amount of the transaction. Each transaction can only have a single value. By default, this value equals the **MW Blocks for Transactions** option that is specified on the *Simulation Tab* of the PowerWorld Simulator Options Dialog.

### Ramping (min)

The length of time over which the transaction will increase from zero to its scheduled amount. By default, this value equals the **Transaction Ramp Time** option that is specified on the *Simulation Tab* of the PowerWorld Simulator Options Dialog.

### Transaction Type

Designates the transaction as either non-firm (recallable) or firm (non-recallable). Simulator currently does not use these identifiers in any of its calculations. They are provided for your bookkeeping.

### Buyer's Price (\$/MWhr)

Price the buyer is paying for the transaction in \$/MWhr. The seller is receiving this amount, minus any wheeling charges. Modifying the **Buyer's Price** automatically changes the **Seller's Price** by a corresponding amount.

### Wheeling Price (\$/MWhr)

Indicates the total paid to areas that wheel power between the two transacting parties. This field is visible only if you have defined wheeling areas using the Wheeling Areas Dialog.

### Seller's Price (\$/MWhr)

Price the seller is receiving for the transaction in \$/MWhr. Modifying the **Seller's Price** automatically changes the **Buyer's Price** by a corresponding amount.

### Start Time, End Time

Starting and ending time for the transaction, entered in the form of hh:mm.

### Start Day, End Day

The start and end days for the transaction. These values are obeyed only when **Day Type** is *Specified Days*.

### View/Modify Wheeling Areas

Pressing this button brings up the Transaction Wheeling Dialog, which can be used to identify those areas that will provide and bill transmission service for the transaction. Such areas are called *wheelers*.

### Day/Time Type

Specifies the days on which the transaction will take place. Select *Specified Days* if the transaction should be modeled only on the days you have specified using the **Start Day** and **End Day** fields. Select *Every Day* if the transaction should occur between **Start Time** and **End Time** every day of the week, including weekends. Select *Week Days* if the transaction should occur between **Start Time** and **End Time** only on days Monday through Friday. Select *Weekends* if the transaction should occur between **Start Time** and **End Time** only on Saturdays and Sundays. Finally, select **Always On** if the transaction should be modeled at all times during the simulation.

**Transaction Enabled**

This box will be checked if the defined transaction is currently being used for the simulation.

## Wheeling Areas Dialog

When two areas participate in a power transaction, the power that is exchanged likely will flow through one or more neighboring areas that are not directly participating in the transaction. If a significant portion of the power flows through a particular entity, that entity likely will try to bill the transacting parties for transmission service. Such an entity is called a *wheeler*. The **Wheeling Areas Dialog** allows you to identify areas that will wheel power as part of the transaction, and at what cost.

You may invoke the Wheeling Areas Dialog from the MW Transaction Information Dialog by clicking on the **View/Modify Wheeling Areas** button.

The table that occupies the left half of the Wheeling Areas Dialog identifies each wheeling area that has been defined for the transaction. This table is a Case Information Display and thus exhibits properties and controls common to all case information displays.

To define a new wheeling area for a transaction, specify an area name (or an area number) and click the **Add New Area By Name** (or the **Add New Areas By Number**) button. If you choose to identify new wheeling areas by number, you may add more than one new wheeling area at a time by specifying a range of area numbers. New wheeling areas are added to the Wheeling Areas in Transaction table, with the wheeling cost set equal initially to a default value of \$2/MWHR. You can change the wheeling charge by specifying a new value in the grid.

To delete a wheeling area, right-click on the row of the record you wish to delete and select *Delete* from the resulting local menu.

To abort the wheeling area definition process without saving changes, click **Cancel**. To save changes and close the dialog, click **OK**.

## Limit Violations Displays

### Limit Violations

The **Limit Violations Dialog** allows you to see buses, lines, or interfaces that exceed their limit. The following fields are shown on the display:

**Number of Line/Transformer Violations**

Shows the total number of violations reported on **Line and Transformer Limit Display**. This field is read-only.

**Number of Bus Voltage Magnitude Violations**

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.

**Number of 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 that are defined. For more information see Limit Monitoring Settings.

**Elements To Show**

Change this value to modify which elements are displayed in the Buses, Lines, and Interfaces. Set it to...

<b>All Elements</b>	to show all Buses, Lines and Interfaces regardless of monitoring settings.
<b>Monitored Elements</b>	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.
<b>Violating Elements</b>	to only display Buses, Lines and Interfaces that are violating. See Limit Monitoring Settings for more information on setting what is considered a violation.

**Buses, Lines and Interfaces Tabs**

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.

The default fields for each tab are as follows:

**Buses Tab****Number, Name**

Bus number between 1 and 99,999, and its alphanumeric identifier.

**Area Name**

Alphanumeric identifier of the bus' area.

**Monitor**

Specifies whether the individual element is eligible for monitoring. See Limit Monitoring Settings for more information.

**Limit Group**

The limit group the element belongs to. See Limit Monitoring Settings for more information.

**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.

**Lines Tab****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.

**Max Flow**

If the **Treat Transmission Line Limits As Equivalent Amps** option is selected, then the **Max Flow** field indicates MVA loading for transformers and amp loading for lines. The value is the maximum flow of the “from and “to ends of the device. If the option is not selected, then MVA flows are specified for both transformers and transmission lines.

**Monitor**

Specifies whether the individual element is eligible for monitoring. See Limit Monitoring Settings for more information.

**Limit Group**

The limit group the element belongs to. See Limit Monitoring Settings for more information.

**Max Percent, Limit**

Percentage loading of the transmission line or transformer at the end with the higher flow, and its MVA or Amp limit. The Percent field is only valid if the MVA limit is nonzero and is calculated as  $\text{flow/limit} * 100$ .

**MVA or Amps?**

Units used with the Max Flow and the Limit field. All flows are expressed either in MVA or amps.

**Interfaces Tab**

**Name**

Alphanumeric identifier for the interface (24 characters maximum).

**Monitor**

Specifies whether the individual element is eligible for monitoring. See Limit Monitoring Settings for more information.

**Limit Group**

The limit group the element belongs to. See Limit Monitoring Settings for more information.

**MW Flow**

Current MW flow on the interface.

**MW Limit**

Interface limit in MW.

**Percent**

Percentage loading of the interface, calculated as  $\text{MW Flow/MW Limit} * 100$ .

## Limit Monitoring Settings

Simulator offers many tools to study the capabilities of a power system. Examples include:

- Limit Violations Display
- Contingency Analysis
- Available Transfer Capability
- Optimal Power Flow
- Voltage Adequacy and Security 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 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 tabs on the Limit Monitoring Settings Dialog allow you to change these settings.

**Buses, Lines, and Interfaces Tabs**

Each tab shows a list of the respective type of power system element. The important columns include

**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

Note: If **Use Area/Zone Filters on List Displays** is checked then the Area/Zone Filters will effect what is displayed on these lists.

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.

**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 elements belongs to a single Limit Group. By default, all power system elements are in the same Limit Group that is named “**Default**”.

The limit group stores important values regarding the enforcement and monitoring of the power system elements within the Limit Group. The values are

**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.

**Line/Transformer Percentage:** The percentage to which Simulator's study tools will limit a line. 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/Transformer Rating Set:** You may define three different ratings to transmission lines or transformers. These ratings are called Rating A, B and C (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 during Contingency Analysis.

**Treat 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

$$\text{Amp Rating} = \frac{\text{MVA Rating}}{\sqrt{3} * \text{Base kV}} * 1000$$

**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 Rating Set:** You may define three different ratings to an interface. These ratings are called Rating A, B and C (for more information see Line/Transformer Information). 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.

**Low Voltage Limit:** Buses will be flagged as violated if they fall below this per unit voltage.

**High Voltage Limit:** Buses will be flagged as violated if they go above this per unit voltage.

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.

**Elements To Show**

Set to Monitored Elements to only display Buses, Lines, and Interfaces that meet the conditions for monitoring.

Set to All Elements to show all Buses, Lines and Interfaces regardless of monitoring settings.

**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.

#### Limit Group Values

The values regarding the Limit Groups are graphically displayed here to make them easier to modify.

#### Do not monitor radial lines and buses

Check this box to ignore limits on radial lines and buses throughout the case.

## Ownership Displays

### Ownership

Buses, loads, generators and lines may have an associated owner or owners, along with the percentage of ownership for each owner. Buses and loads can have one owner, while generators and lines can have up to 4 owners.

The primary source of owner information for case files is from PTI raw data. However, the file needs to be in version 25 format for owner data to be present. If the file is not a version 25 format file, the owner data is initialized to no owners.

To view the different owners and see the number of objects they own, select **Case Information, Other, Ownership**. For more information, see Owner Data Information Display.

When you save the case, any changed values are saved in the .pwb file, either in the Owners section of the .pwb or with the individual objects. Owner data can also be saved with PTI version 25 and 26 text formats.

### 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. You can right click on the owner grid to open the local menu, from which you can perform various actions, including inserting or deleting owners, and showing an informational dialog for a specific owner. Once an owner dialog is displayed for a particular owner, you can browse through information on all the owners.

### Owner Dialog

The dialog for an individual owner displays each of the elements for that owner, along with the percentage of ownership. By default, the owner dialog displays only the elements for the selected owner. However, it is possible to display all objects in the case in the owner dialog by pressing the Show All Objects In Case button. Pressing the same button again will change the dialog back to showing only the objects for the selected owner.

A device can be added for an owner by using the edit boxes at the top of the different tab pages. Generators and loads are based on the device number only, and multiple objects can be added to the owner at once by using commas and dashes to specify ranges of bus numbers, for example buses 4 and 6 could be 4,6 and buses 4 through 6 could be 4-6. Generators and lines have to be entered in the edit boxes one at a time, specifying bus numbers and identifiers. Using the edit boxes will call up the Update Owners Dialog, in which the user must redistribute the percentage ownership to include the added owner. If any owner percentage in the Update Owners dialog is left 0, that owner will be removed from the owner list for the device.

## 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 Update Owners. See Update Owners Dialog for more information.

## 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%. The make changes to the owners and ownership percentages for a generator, right-click on the grid and choose Update Owners. See Update Owners Dialog for more information.

## Update Owners Dialog

The Update Owners dialog is only used for updating the owner information of generators and lines. Currently, Simulator supports up to four owners for lines and generators. 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: it is possible to have the ownership percentage be less than 100%, but you cannot have a combined percentage greater than 100% for any device.

## Load Flow Displays

### 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.

However, Excel does have limitations on the number of rows and columns that could quickly be exceeded with a Jacobian matrix from a large case.

## 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: Injection Groups

This chapter describes the setup and use of injection groups. Injection groups are useful for modeling generators and loads together like a unit.

The following topics are covered in this chapter:

- Injection Group Overview
- Creating and Deleting Injection Groups
- Injection Group Display and Dialog
- Participation Points Overview
- Participation Point Record Display
- Adding Participation Points

### Injection Group Overview

An injection group is a collection of loads and generators. In that respect, injection groups are similar to areas and zones, which also group generators and loads together according to the buses to which the generators and loads are attached. However, unlike with areas and zones, a generator and load can belong to more than one injection group. Moreover, a single injection group may contain generators and loads from several different areas and zones. Thus, injection groups are useful when you need to model a collection of generators and loads 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. They are called **injection groups** because their components (generators and loads) are objects that inject power into the network.

### Working with Injection Groups

#### Creating Injection Groups

To create an injection group, select **Case Information, Injection Groups** from the main menu to open the Injection Group Case Information Display. Click the right mouse button on this display, and select **Insert** from the resulting local menu. Specify a name of up to 24 characters to identify the new injection group, or click the **Load** button to import injection groups from an Injection Group Auxiliary Data File. If you specify an injection group name, the Injection Group Dialog will open, which will allow you to add injection points to the group.

Alternatively, if the Injection Group Dialog is already open, you can click the button labeled **New**. You will then be asked to supply the name of the group you wish to add.

#### Deleting Injection Groups

To delete an injection group, select **Case Information, Injection Groups** from the main menu to open the Injection Group Case Information Display. Click the right mouse button on this display, and select **Delete** from the resulting local menu.

Alternatively, if the Injection Group Dialog is open, you can delete the injection group listed in the Name dropdown box by clicking the **Delete** button.

## Injection Group Display

The Injection Group Records Display presents data describing each injection group in the case. It 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 Injection Group Dialog. You can also sort the injection group records by clicking on the heading of the field by which you want to sort.

To show the Injection Group Records Display, select **Case Information, Injection Groups**

By default, the Injection Group Records 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.

### **% Gen PF**

Indicates the degree to which generators will contribute to the output of the injection group relative to loads. An injection group that has a % Gen PF value of 100.0% receives all of its output from generator points; an injection group that has a % Gen PF of 50% and a % Load PF of 50% receives equal contributions from its constituent loads and generators.

### **Number of Loads**

Identifies the number of loads contained in the injection group.

### **% Load PF**

Indicates the degree to which loads will contribute to the output of the injection group relative to generators. An injection group that has a % Load PF value of 100.0% receives all of its output from load points; an injection group that has a % Load PF of 50% and a % Gen PF of 50% receives equal contributions from its constituent loads and generators.

## Injection Group Dialog

The Injection Group Dialog provides information about injection groups and allows you to modify them. Specifically, the Injection Group Dialog lists the number of generators and loads contained in the group and the percentage contribution of generators and loads to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which you can add and delete points from the injection group's list of participants and change various attributes of the points.

To view the Injection Group Dialog for a particular injection group, open the Injection Group Records display by selecting **Case Information, Injection Groups** from the main menu. Find the injection group in which you are interested, and right-click 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. You can select a different name from this dropdown box to display information for another injection group.

### **New, Delete**

To insert a new injection group from this dialog, press **New**, and supply the name you want to give the new injection group. To delete the injection group that is currently being shown, click **Delete**.

#### # Gens

Displays the number of generators contained in the injection group.

#### # Loads

Displays the number of loads contained in the injection group.

#### % Gen Part., % Load Part.

Displays the relative contributions of generators and loads to the output of the injection group.

#### Elements

The first tab on the right hand side of the Injection Group Dialog lists the subsystem elements that make up an injection group. The subsystem elements can be either Areas or Buses. The two tabs at the bottom of the Elements page allow you to switch between viewing the Bus elements and the Area elements. To add new buses or areas, go to the proper tab of the Elements page, and use the **Add New ... by Name** or **Add New ... by Number** to add the elements of choice to the subsystem.

#### Participation Points

The second tab on the right hand side of the Injection Group Dialog lists the points that make up the injection group. This display is called the Participation Point Record Display. By right-clicking on this display, you can add and delete points from the injection group, and you can change the properties of specific points.

## Injection Group Contributions By Area

Injection groups may include generator and load points from an arbitrary number of control areas. By default, the contribution of each generator or load point is controlled just by the point's participation factor. Sometimes you may need more precise control over how much each generator or load contributes to an injection group's export in order to achieve a certain percentage contribution from each area represented in the group. For example, suppose the injection group consists of points (generators and loads) contained in areas A, B, and C. Using the **Injection Group Contributions By Area Dialog**, you can scale the factors for each of these points such that, for example, 40% of the total contribution comes from points in area A, 30% from points in area B, and 30% from points in area C. Within each area, the points contribute in proportion to the original participation factors you defined. However, the total contribution from each area is determined by the percentages you enter on the **Injection Group Contributions By Area Dialog**.

The dialog houses a table containing the following fields:

<b>Area Number</b>	The number of the area
<b>Area Name</b>	The name of the area
<b>% Contribution</b>	The current percent contribution of the area. This is calculated as the sum of the participation factors of the injection group's points that belong to the area divided by the sum of all the injection group's points' participation factors.
<b>New % Contribution</b>	The new percent contribution you want to achieve from the area. The participation factors of the points that belong to the area will be scaled so that the area's total participation becomes this percentage of the injection group's total participation.

To set the contribution for a specific area, simply type in a new value for the **New % Contribution** field for that area. This is the only field you can change in the table. Only areas that are represented in the injection group by load or generator points will be listed in the table.

After you have set the new percent contributions to your liking, click the **Change Contributions By Area** button. If you changed your mind and do not want to rescale the participation factors, click Cancel.

## Participation Groups

### Participation Points Overview

A participation point is a member of an injection group. It is a generator or load that participates in, or contributes to, the output of an injection group. Each participation point record identifies the generator or load 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).

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 you to add or delete points or to find out more information about them using the Participation Point Dialog.

### Participation Point Records Display

The Participation Point Records display shows information about the points that comprise a particular injection group. You can access this display for a particular injection group only from the Injection Group Dialog.

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 you can print, copy, and modify its records as well as view the Generator Dialog or Load Dialog for a particular point. It also allows you to add participation points to the group through the Add Participation Points Dialog, and to delete participation points. You can also sort the participation point records by clicking on the heading of the field by which you want to sort.

The Participation Point Records display shows the following fields by default:

**Point Type**

Every participation point is either a generator (GEN) or load (LOAD).

**Number**

Identifies the number of the bus to which the generator or load is connected.

**Name**

Identifies the name of the base to which the generator or load is connected.

**ID**

Identifies the ID of the generator or load.

**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

<b>SPECIFIED</b>	The participation factor was specified as a constant.
<b>MAX GEN INC</b>	The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.
<b>MAX GEN DEC</b>	The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.
<b>MAX GEN MW</b>	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:

<b>SPECIFIED</b>	The participation factor was specified as a constant.
<b>LOAD MW</b>	The participation factor was defined as the size of the load in MW.

The Initial Value field is important if you intend to save the injection groups to an auxiliary file and use them with another case that might have a different generation dispatch or load profile. If you specify the Initial Value for a point as GEN MAX INC, for example, and load the point 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 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 you toggle the value of the Initial Value field, the point's participation factor, shown in field ParFac, will update to match the new definition. The Initial Value field must equal SPECIFIED for you want to change the value of the point's ParFac field by typing the new value directly into the field.

If AutoCalc is NO and you do not intend to use the injection group with any other case, then you should either ignore this field or set its value 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 or generation change for 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.

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 you to add participation points to an injection group. You can access this dialog from the Participation Point Records display by right-clicking on that display and selecting **Insert** from the resulting local menu.

The Add Participation Points Dialog features three tabs, one for adding generator points, another for adding load points, and a third for adding points from groups that have already been defined. The three tabs are almost identical and contain the following controls:

### Use Area/Zone Filters

If this box is checked, the list box beneath it, which lists generators or loads depending on the tab, will list only those elements contained in areas or zones whose area/zone filter setting is YES. If this box is not checked, all generators or loads in the case will be listed.

### Element List

The box that occupies the left side of each tab lists the generators, loads, or groups (depending on which tab is active) that can be added to the injection group. You can select multiple elements from each of these lists. To select several elements in a row, drag the mouse to highlight the elements you want to add. Alternatively, click the first element you want to add, press and hold the shift key, and click the last element you want to add. To select elements that are not adjacent in the list, click the first element you want to add and hold down the CTRL key while clicking the other elements you want to add.

Once an element is selected in the Element List, it is ready to be added to the injection group.

### Participation Factors

You have several options for how to define the participation factor of the points you have selected. For generators, you can specify a value. Use the generator's present participation factor (which comes from the case and is displayed in the Generator Records Display), or calculate the participation factor as the difference between its maximum output and its present output or its maximum output capability. For loads, you can specify a value, or you can base the participation factor on the load's size. For groups, you can specify a value to use for every point in the injection group, or you can use the values already defined for the injection group.

### Recalculate Factors Dynamically

If this box is checked, the participation factors of the points you are adding 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 you are adding will be fixed at the values defined at the time they were created.

To add the points that you have selected, click the **Arrow** button. The new points will be added to the list box on the right.

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 **Trash** button.

To close this dialog, click **OK**.

## Chapter 10: 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:

- PowerWorld Solution/Environment Options
- Controlling Simulations and Solving the Power Flow
- Description of Area Control and Transaction Modeling
- Charts
- Bus View Display
- Substation View Display

## Options

### Solution/Environment Options

Simulator provides a flexible environment for simulating power system operation by offering you access to a number of customizable options. The **PowerWorld Simulator Options Dialog** houses six pages of options that you can customize to tailor the simulation to your needs. To display this dialog, select **Options/Tools, Solution/Environment** from the main menu. 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 six tabs of options for the Solution/Environment Options dialog:

Power Flow Solution Options

Environment Options

Case Information Display Options

Template Options

Simulation Options

Limits Options

### Power Flow Solution Options

The Power Flow Solution Tab offers various options regarding how Simulator solves the power flow problem. There are four sub-categories on the Power Flow Solutions tab; General, Automatic Controls, Area Interchange, and Storage. Each of these categories is shown on tabs at the bottom of the page. Many of the options in the Power Flow Solution options will be of interest only to advanced users of the package.

## Power Flow Solution: General

### Initialize From Flat Start Value

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.

### Allow Multiple Islands

If checked, multiple islands are allowed during the solution.

### Disable Power Flow Optimal Multiplier

If checked, the Newton solution process will ignore the optimal multiplier. The optimal multiplier is a mathematically calculated value that indicates the ability of the iterative solution process to continue towards a valid solution. If the optimal multiplier becomes too small, it signifies that the solution process has hit a point where it is changing by extremely small amounts during each iteration. 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.

### Restore Initial State on Restart

If checked, a paused simulation will revert to initial conditions whenever the user selects **Simulation, Play** from the main menu to start a simulation, or selects **Simulation, Reset** followed by the Play option to restart a simulation. Otherwise, selecting Play with a paused simulation will simply resume the paused simulation.

### MVA Convergence Tolerance

The MVA convergence tolerance serves as a yardstick for determining when 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.

### 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.

### 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.

#### **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 Full Newton Power Flow Solution or an optimal power flow (OPF) solution.

## Power Flow Solution: Automatic Controls

#### **Disable Checking Gen MVR Limits**

If checked, the MVR limits are ignored for all the generators in the case during a Power Flow Solution. By default, this option is not selected.

#### **Check Immediately**

If checked, violations of the MVR limits for each generator are checked before the Power Flow Solution is started. Normally, the Power Flow Solution is started without checking the MVR limits until after a solution has been reached. If any of the MVR limits are violated after the solution is reached, the violations are corrected, and the power flow is solved again. With this option checked, the violations are checked before the first time the power flow is run. By default, this option is not selected.

#### **Disable Switched Shunt 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.

#### **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 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.

#### **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.

#### **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.

#### **Prevent Controller Oscillations**

Sometimes, a power flow will fail to converge because certain automatic controls such as MVR 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.

## Power Flow Solution: Area Interchange

### **Disable Area Interchange Control**

If checked, the enforcement of the MW interchange constraints for all areas is disabled. By default, this option is not checked.

### **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.

## Power Flow Solution: Storage

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.

## Environment Options

The Environment Options Tab 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 online displays. These options include:

### **Disable Showing Blackouts**

You can dramatize a power flow case's failure to converge by representing it as a blackout. The background of the online 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 "blacked out, or shed, to restore the system to a viable operating state. Thus, displaying the failure to converge as a blackout has physical significance.

#### **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 oneline diagram, Simulator will store the contour with the oneline diagram when you save the case or save the oneline.

#### **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 of the Program Palette or one of the two Power Flow Solution options available from the **Simulation** menu on the main menu.

#### **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.

#### **Auto Load Script File**

If checked, the script file is automatically loaded with the case. This is not checked by default. Note: scripts can be loaded manually from the option in the simulation menu.

#### **Minimum Screen Font Size**

The minimum font size at which text is visible on the screen. 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.

#### **Recently Used File List Entries**

The maximum number of file names and locations stored in the History List of the File menu.

#### **Power Units for Display**

Allows switching between displaying power values in Mega- units or Kilo- units.

#### **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*.

#### **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.

### 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.

### Main Online File

This option is used to identify the primary oneline diagram to use with the case. The main oneline is the file that is displayed when you first open the case. The dropdown box lists all the oneline files that reside in the same directory as the case. Select one of these files, or enter the full path of the oneline 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 oneline diagram file if it cannot find a oneline diagram file for the case you are trying to open. For example, there is no oneline 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 oneline file whenever you open a PSS/E raw file, check the **Use Default Online File** to have Simulator open the oneline identified in the **Default Online File** box whenever it encounters a case that has no associated oneline. The default oneline must exist in the same directory as the case you are trying to open.

### Automatically Load Contingencies when Case is Opened

If checked, Simulator will open the file specified in the Contingency File field when the case is opened and load the contingencies from the file into memory.

## Case Information Display Options

PowerWorld uses numerous case information displays to show power system data 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 Enter Color** 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 Toggle Color** 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 Limit Color** button.

### Default Row Height

This option sets the 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 20.

### Disable Auto Refresh of Case Information Display

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.

### 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.

#### **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.

#### **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 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.

## Template Options

Simulator offers the ability to apply case *templates* to open power flow cases. Templates are binary files that store such things as schedules, case information display customizations, and settings for many of the options described in this section. For more information on how to work with templates, please see Using Case and Online Templates.

#### **Save in Template File**

The type of information to store in the case template. Your choices include schedules, case information customizations, and case options, as configured from this PowerWorld Simulator Options Dialog. In addition, a default auxiliary data file can be saved with the case template, allowing the default auxiliary file to be automatically applied to a case whenever the template is applied.

#### **Use Default Case Template**

By checking this option, you can also have Simulator load in a default set of options, regardless of the case you are working with. If this option is checked, whenever Simulator tries to open a new case, it will apply the options stored in the default template file you identify. The template file must exist in the directory of the case you are attempting to open.

## Simulation Options

The Simulation Tab controls such things as the timing of the simulation and the structure of energy transactions. The Simulation Tab offers the following options:

#### **Simulation Never Ends**

If checked, this option forces the continual simulation of the load flow when using the Play functionality, ignoring the Simulation Start and Simulation End Time settings.

#### **Simulation Start and Simulation End Times**

The start and end times of the simulation have both time and day attributes, since Simulator can model a system over multiple days. All time specifications must have the form hh:mm. Weeks are assumed to begin on Mondays.

#### **Current Time**

This indicates the current time of the simulation. Adjust this value to rewind or fast-forward the simulation to a particular point in time.

### **Simulation Speed-Up**

The Simulation Speed-Up controls how fast the simulation runs relative to actual time. For example, entering 60 in this field causes the simulation to run at 60 times actual time so that one minute of simulation time occurs in one second of actual time. This field is visible only if the **Fixed Time Step** box is unchecked.

### **Fixed Time Step**

By using this option you can specify the rate of the simulation in terms of the number of seconds to advance with each Power Flow Solution. If this box is checked, the **Simulation Speed-Up** field is relabeled the **Fixed Time Step (seconds)** field.

### **Fixed Time Step (seconds)**

The Fixed Time Step (seconds) field is used to specify the number of seconds the simulation should advance with each Power Flow Solution. For example, specifying a fixed time step of 60 seconds means that the simulation time will advance 1 minute each time the power flow is solved. This field is visible only when the **Fixed Time Step** box is checked.

### **Default Area Number**

This option specifies the default area number used with the strip-chart displays.

### **MW Blocks for Transactions**

This option indicates the default size of new MW transactions. When you try to insert new MW transactions using the MW Transactions Display and MW Transactions Dialog, the initial transaction amount will default to this value.

### **Transaction Ramp Time (seconds)**

This option sets the default ramp rate for inter-area transactions. This value specifies the amount of time in seconds over which a transaction will ramp from zero to the level indicated by the **MW Blocks for Transactions** field. When you try to insert new MW transactions using the MW Transactions Display and MW Transactions Dialog, the initial value of the **Ramping** parameter will default to this value.

### **Scheduled System Frequency**

Indicates the scheduled system frequency in Hz.

### **Cost of Unserved Energy**

This field is used to assign a penalty cost to areas that fail to fulfill their side of energy transactions. This cost will be included in computing the total cost of operating the offending area.

### **Case Load Multiplier**

This field is used to specify a scalar by which to multiply the value of all real and reactive loads throughout the system. See Load Modeling for further details.

### **Transformer Tap Delay**

Typically, a time delay is built into the switching of transformer taps. You can model this delay during timed simulations in Simulator by specifying a nonzero value for the Transformer Tap Delay in seconds. If this value is nonzero, Simulator will wait until the specified delay time has elapsed before switching the transformer taps as long as the **Use Transformer Tap Delay** box is checked.

### **Use Transformer Tap Delays**

This box must be checked in order for the tap delay setting in the **Transformer Tap Delay** box to take effect.

#### Unexpected Events

This field allows you control the frequency with which “unexpected events, such as lightning hitting a transmission line, can occur during a simulation. When such an event occurs, the line’s circuit breakers immediately open to protect the line from further damage. For some events (such as a lightning strike), you can immediately re-close the line (i.e., you can use the oneline to close the circuit breakers). However, for other events, like tornadoes, the line will stay open for the remainder of the simulation (since it can take months to rebuild a line). These events occur at random times. Whenever an unexpected event occurs, you will be shown a message box, which you must acknowledge to proceed.

#### Auto Start

If the Auto Start box is checked, a timed simulation for the case will be launched as soon as the case is loaded into memory.

#### Auto Solve on Load

If this option is checked, the case is automatically solved when it is loaded, **provided Simulator is in Run Mode**; otherwise, the case is not solved until the user starts the simulation.

## Limits Options

#### 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 Analogs of Objects with Limit Violations

If checked, the text 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.

## Popup Hints Options

One of the novel features of Simulator oneline diagrams is their ability to display popup hints. A popup hint is a window that appears over a oneline when a user places the mouse cursor over a power system component and leaves it there for a short duration. After the pause, the popup hint window appears to convey information about that component. The window will disappear once you move the mouse.

Naturally, you may want to show different information in the popup window for each different type of power system component. The **Popup Hints** tab of the Solution / Environment Options Dialog allows you to customize the content of popup windows for each category of power system component.

#### Disable Showing Popup Hints

When this box is checked, the tab looks rather plain, as all other controls are hidden. Uncheck the box to enable the display of popup hint windows.

#### Filters

To limit the display of popup hint windows to a subset of all buses, select a value from the **Filters** selection group. To place no additional restrictions on the components for which popup hints are displayed, select **Show for all**. To limit the display of popup hint windows to a particular limit group, select **Show for Limit Group** and designate the limit group for which you want popup windows to be displayed. To limit the display of popup hint windows according to the Area/Zone Filters or a particular advanced filter, select **Show for Filters**, and select the name of the filter to use to control the display of popup hints. To enforce the use of the Area/Zone –based display filters, select “**Area / Zone** from the list of filters instead of a particular named filter.

If you have chosen to display popup hints, the rest of the Popup Hints tab will now be populated with controls that help you customize the content of the popup hint windows.

#### Object Type

This box lists the different types of power system objects for which Simulator can display popup hints. These include buses, generators, loads, switched shunts, areas, zones, and interfaces. Different configuration settings are maintained for each type of object. Select the type of object whose popup hint settings you currently want to manage.

#### Show Popup Hints for This Object Type

If this box is unchecked, Simulator will not generate popup hint windows for any object of the currently selected type. Check this box to ensure that Simulator will display popup hint windows for the selected type of object. This will also provide you access to the rest of the controls on the tab.

The rest of the tools on the tab give you control of the content of the popup hint windows. The heart of this interface is a text window that enables you to format the content and appearance of the popup hints for the currently selected object. Any text that you type into this window that does not appear inside the structure <FIELD= ... > will be displayed verbatim in the popup hints generated by Simulator. To add the display of a specific attribute or characteristic of an object to the popup hint, position the cursor in the text window at the position where you would like the value to be displayed. Then, do either of the following:

- Select the name of the attribute from the Object attributes dropdown list. This will insert the text <FIELD=“fieldname”> into the text window, where field\_name describes the attribute you just selected. Or
- If you know the fieldname of the particular characteristic you wish to include, directly type <FIELD=“fieldname”> into the text window.

The second option also suggests how fields may be removed from the popup window display: simply delete the <FIELD=...> tag that corresponds to the characteristic you wish to remove.

The settings you have selected will be saved when you click the **OK** button to close the Solution / Environment Options Dialog. If you do not wish to save the settings, close the dialog by clicking the **Cancel** button.

The options you specify on the Popup Hints tab are stored in the system registry. They are not saved with the power flow model or the oneline diagram.

## Solution and Control

### Simulation Control

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation may focus on a single instant of time or may chart the evolution of the system over time using a series of system snapshots. The Simulation Menu and the

corresponding Program Palette are used to control both types of simulation. For timed simulations, the menu is used to start/continue the simulation, pause the simulation and reset the simulation to its starting parameters. The system clock is usually visible during a timed simulation and shows the current simulation time along with the simulation's start and end time. The system clock will appear either in its own window or as a series of panels in the right corner of the window's status bar. The other type of simulation, that of a single instant of time, is accomplished by selecting **Single Solution** from either the main menu or the Program Palette. This activity performs a single Power Flow Solution. See Solving the Power Flow for more information.

The following tasks are available from the Simulation Control menu and the Program Palette:

#### Starting a Timed Simulation

To commence a timed simulation, select **Simulation, Play** from the main menu, or click the play button on the Run Mode Palette. The simulation clock, visible in either its own window or the status bar, will keep track of the simulation time. Regardless of the speed of your computer, the simulation time will change at a fixed rate that is governed by the Simulation Speedup selected in the Simulation Options Dialog.

You can reset the simulation at any time by choosing the **Simulation, Restart** command, or by clicking the restart button on the Run Mode Palette.

#### Pausing a Timed Simulation

Once the simulation has started, you can pause it at any time by selecting **Simulation, Pause** from the main menu or by clicking the pause button on the Run Mode Palette. It is helpful to pause the simulation when you want to take a closer look at a particular instant of the simulation.

#### Continuing a Timed Simulation

Once the simulation has been paused, you can continue with the simulation by selecting **Simulation, Play** from the main menu, or by clicking the play button on the Run Mode Palette.

#### Restoring a Previous Solution or 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 **Simulation, Restore, Last Successful Solution** 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 **Simulation, Restore, State Before Last Solution Attempt** from the main menu. 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 **Options/Tools, Solution/Environment...** to open the Solution/Environment 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.

### Performing a Single Solution

Whenever a timed simulation isn't currently active, you can instruct Simulator to perform a single power flow calculation by selecting **Simulation, Single Solution - Full Newton** from the main menu or by clicking **Single Solution** on the Program Palette. A single solution solves the power system using the specified solution technique. The simulation time and total system costs do not change as a result of the single solution.

### Reset to Flat Start

Select **Simulation, Reset to Flat Start** from the main menu 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 Simulation and Environment Options Dialog to initialize every solution from a flat start.

## 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. When Simulator performs a timed simulation, it actually performs a sequence of power flow calculations, each based on data that present a snapshot of system conditions at a particular instant. Since it is based on power flow, Simulator models the system as being of constant frequency. This assumption is only an approximation, but frequency deviations are generally very small in everyday operation.

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.

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.

## 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. Normally, this requirement is 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

$$\text{ACE} = \text{Pactual} - \text{Pscheduled} + (\text{a term dependent upon system frequency})$$

where Pactual is the actual amount of MW flowing out of an area. If power is actually flowing into the area, Pactual is negative. Pscheduled is the amount of power scheduled to flow out of the area and thus equals the areas total sales minus its total purchases, both expressed in MW. The last term in the ACE depends upon the deviation of the actual system frequency from the scheduled system frequency. When the system frequency is modeled using constant frequency, this term is always equal to zero. Otherwise, the equation assumes uniform, but not constant, frequency throughout the system. In this case, this term, known as the *frequency bias*, equals  $-10 B \Delta f$ , where B is an area specific bias factor with units of MW/0.1 Hz (always negative), and  $\Delta f$  is the deviation of the frequency from the nominal frequency (usually either 50 Hz or 60 Hz). Simulator currently always assumes a constant frequency.

In Simulator, Pscheduled has two components, *Scheduled Transactions* and *Base Transactions*. The two types are provided in order to simplify transaction modeling in some studies. Scheduled Transactions have a specified starting time, ending time, MW amount, and price. However, in many power flow studies, in which one is concerned only with the static flow of power in the transmission network and not specifically with cost information, it is much easier just to model the net flow of power from one area to another (or even to an unspecified area). For these situations, there is no need to specify price or start and end times. Simulator refers to these as Base Transactions. Both Base and Scheduled Transactions can be defined on the Area Information Dialog.

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 five 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.

**Participation Factor Control**

The output of all AGC generators in the area 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, which, when pressed, opens the Generator Participation Factors Dialog.

**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.

**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.

**Area Slack Bus Control**

Only the output of the area's slack bus changes automatically to drive the area control error (ACE) to zero.

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.

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. 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 ( $\lambda_i$ ) are equal. This value is known as the system  $\lambda$  (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.

## 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 area's 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. The Area Information Dialog has a button labeled **Set Factors** that is enabled only if the Participation Factors 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 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.

<b>Difference Between Max and Current Output</b>	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.
<b>Constant Value of</b>	The participation factor for each generator is set to the same hard-coded value.
<b>File</b>	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.

## Transactions

### Area Transaction Modeling

One of the primary benefits of interconnected power system operation is the ability to do power transactions between areas. In its simplest form, a power transaction entails one area selling a specific amount of power for a time period (often specific, sometimes not) at a specific price to another area.

Simulator offers two complimentary mechanisms for specifying area interchange: Base Area Interchange and Scheduled Area Interchange. These mechanisms may be used simultaneously, provided you understand that the use of base transactions in economic studies can skew results considerably. This discussion should clarify the issues involved in using the two types of transactions.

#### Base Area Interchange

The base interchange for an area is the amount of power (in MW) scheduled to be exported from the area to either another area or to an unspecified buyer(s). However, since base interchange is intended only for use in standalone power flow studies, no start/end times and no price are associated with the transaction. Since the price and sometimes the buyers **are not known**, the use of base interchange is **DEFINITELY NOT RECOMMENDED** when you are doing pricing studies/simulations. Since no price is specified, from an economic modeling point of view the area is essentially giving the power away for free. This can, of course, have a major impact on area economics. Additionally, base interchange has no starting or stopping time; it is always assumed to be active.

However, the use of base interchange is convenient when you are simply doing power flow studies/simulations (i.e. without economic considerations). Base interchange values can be set on the Area Dialog. Please note that when you use base interchange with unspecified buyers, it is your responsibility to ensure that the net of all base transactions add to zero. That is, in order for some areas to export, other areas must be importing a corresponding amount. Base interchange values can also be viewed and modified on the onelines using area fields.

#### Scheduled Area Interchange

The scheduled interchange for an area is the amount of power (in MW) scheduled to be sold from the area to specific buying areas for a specific duration and price. You should use these types of transactions exclusively when you are doing economic/pricing studies/simulations. The only downside to the use of scheduled interchange is that you must specifically set up each such transaction. Please see MW Transactions Display for a description of how to do this.

## Area Transaction Options Dialog

This dialog is outdated in PowerWorld Simulator version 5.0 and later. See help on the Area Information dialog, specifically the Scheduled MW Transactions, for changing, inserting, and deleting scheduled area transactions.

When viewing an area diagram containing area objects created using PowerWorld Simulator version 4.2 or older, a transaction can be defined for an area by left-clicking on the **Deal / Wheel** field in an area's object to open the **Area Transaction Options Dialog**. This dialog allows the user to define a particular type of transaction for the selected area. Note that the dialog does not specify where the resulting transaction is from or to. For either a buying or selling action, the user will need to specify the appropriate transaction for each of the involved areas and the amount of real power (MW) included in the transaction. The costs associated with the transaction can be estimated by pressing the **Estimate Price** button once the MW amount of the transaction has been specified.

## 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 behave in a manner identical with all other 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.

## Charts

### 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 view this display, select **Options/Tools, Charts, ACE Chart** from the main menu, or press the corresponding button on the Simulation Summary tab of the Area Information Dialog. If you use the main menu to view this chart, it shows information for the default area, but this can be changed. 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 over time. To view this display, select **Options/Tools, Charts, Area Load and Generation Chart** from the main menu, or press the corresponding button on the Simulation Summary tab of the Area Information Dialog. If you use the main menu to view this chart, it shows information for the default area, but this can be changed. The strip chart starts to chart 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 (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 over time. To view this display, select **Options/Tools, Charts, Area Losses Chart** from the main menu, or press the corresponding button on the Simulation Summary tab of the Area Information Dialog. If you use the main menu to view this chart, it shows information for the default area, but this can be changed. The strip chart starts to chart 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 (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 over time. To view this display, select **Options/Tools, Charts, Area MW Transactions Chart** from the main menu, or press the corresponding button on the Simulation Summary tab of the Area Information Dialog. If you use the main menu to view this chart, it shows information for the default area, but this can be changed. The strip chart starts to chart 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 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 view this display, select **Options/Tools, Charts, Area Average Cost Chart** from the main menu, or press the corresponding button on the Simulation Summary tab of the area information dialog. If you use the main menu to view this chart, it shows information for the default area, but this can be changed. 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 Oneline

### Bus View Display

The **Bus View Display** feature serves as a graphical analog to the text-based Quick Power Flow List. Like the quick power flow list, the bus view displays enable convenient bus-by-bus navigation through the power system. Moreover, the bus view display contains much the same information as the items on the quick power flow display. 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, as with the quick power flow lists, 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, however, is that you interact with them in the same, familiar way you interact with oneline diagrams. In other words, the bus view displays provide the content and maneuverability of the quick power flow lists with the ease of use of the 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. The next two controls following the *Bus* label allow to specify a bus name (in the first text box) or a bus number (in the second text box). If you type a number or name that does not exist, the bus display will continue displaying the current bus. Following the bus specification fields are two text boxes displaying the name and number of the bus' associated area. These two fields are read-only.

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 black 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. 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. Branches that serve as tie lines are drawn in green for easy identification. Neighboring buses are represented as filled yellow rectangular regions. When you drag the mouse over one of these 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 addition 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. Again, the bus view display is the quick power flow list in the form of a oneline diagram.

The bus view display can be generated using any of the following methods:

- From the main menu, choose **Case Information > Bus View**. You will have to specify a bus name or number upon entry.
- 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 corresponding toolbar icon on the Options/Info Palette.

To switch between the bus view and the main oneline, use the **Window** menu tree on the main menu. To close the bus view display, simply close the form using the **X** button in the top right corner of the bus view display.

## Substation View Oneline

### 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. 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 an addition online 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 object's 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:

- From the main menu, choose **Case Information > Substation View**. You will have to specify a bus name or number upon entry.
- Right click on a substation of interest on the oneline diagram to display the substations local menu, and choose **Substation View**. The substation view display will open with the selected substation already displayed.
- From any of the Substation Records display, right click on a record to bring up its local menu, and choose **Substation View Oneline**. The substation view display will open with the corresponding substation already displayed.
- Click the corresponding toolbar icon on the Options/Info Palette.

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 11: 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:

- Difference Flows
- Generator Economic Curve Displays
- Movie Maker
- Contingency Analysis
- Fault Analysis
- Contouring
- Distribution Factors
- Sensitivities

## General Tools

### Difference Flows

The **Difference Flows** feature provides an easy mechanism for comparing two power system cases. For example, it could 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. Use of this feature affects all aspects of the Simulator environment, for all oneline diagrams, case information displays, and power flow lists will reflect either the operating state of one of the two cases being compared or the differences between those cases.

Using the **Difference Flows** activity is fairly easy:

- Set up a solved power system corresponding to one of the desired operating point. This operating point will be defined as the **Base Case**.
- Select **Options/Tools, Difference Flows** from the main menu, **Difference Flows** from the oneline local menu in Run Mode, or the **Difference Flows** *button* (not the arrow next to the button!) on the Run Mode Palette to display the **Difference Flows Dialog**.
- On the **Difference Flows Dialog**, click the button labeled **Set Present as Base Case**. This stores the current operating point as the *Base Case*. The Base Case is the operating point *against which* the difference flows comparison will be made.
- Define the operating point for which to perform the difference flows comparison. This new operating point, the *Present Case*, will be compared with the Base Case to determine the difference flows. The Present Case may be developed either by modifying the Base Case as desired and re-solving, or by opening a new case using **File, Open Case** from the main menu. In the latter situation, the new case that you open must have the same bus numbering scheme as the Base Case.
- On the Difference Flows Dialog, use the **Case Type to Display** group box to toggle between displaying the Present Case, the Base Case, or the Difference Case, which shows the difference between the Present Case and the Base Case, calculated relative to the Base Case's operating point. The **Case Type to Display** setting controls which operating state - the Base Case, the Present Case, or the Difference Case - is displayed by all oneline diagrams and all

case information displays. Alternatively, click the arrow next to the **Difference Flows** button to produce a list of which case to display: Present Case, Base Case, or Difference Case. The presently selected case is flagged with a checkmark. Select the case you wish to display.

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 **Set Present as Base Case** 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. 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, select **Options/Tools, Present Case Differences from Base Case** from the main menu. The resulting display lists all devices that have been added or removed from the present case relative to the Base Case.

## 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*) and 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 concave 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.

## Movie Maker

Simulator has a feature called Movie Maker that enables you to record the animation of oneline diagrams as AVI movie files. This feature can be useful for producing demonstrations of system conditions and distributing them to colleagues who may not be using PowerWorld Simulator.

To open Movie Maker, select **Options / Tools > Make Movie** from the main menu in Run Mode. A dialog entitled "PW Movie Maker" will open. This dialog allows you to control various settings for the recording of the movie, including its length and screen dimensions. It also provides controls for starting and pausing the recording of the movie, and saving and playing it once the recording has finished.

Note that you must have a PowerWorld oneline diagram open in order to use the Movie Maker. This only makes sense, as without a oneline diagram there are no frames of animation to record.

## Movie Maker Dialog

The PowerWorld Movie Maker Dialog is opened by selecting **Options / Tools > Make Movie** from the main menu in Run Mode. This dialog features the following controls:

#### **Length of Movie**

Two options are provided for specifying the length of the movie to record. You can either use the start recording and pause recording buttons that appear along the bottom of the Movie Maker form, or you can specify that the movie is to contain a fixed number of slides. A slide is simply one frame of animation. If you check the latter option, you must also specify the number of slides to include.

#### **Length of Frame**

The Movie Maker works by capturing sequential frames of animation of a oneline diagram. This particular attribute controls how long each captured frame is kept on the screen during the movie. This duration is expressed in milliseconds.

#### **Dimensions (pixels)**

Controls are provided that allow you to specify the width and the height the movie should assume during playback. During the record process, the oneline image will be scaled to match these dimensions so that the movie will preserve the original resolution when replayed.

Along the bottom of the dialog is a row of control buttons. From left to right, these buttons are:

### **New Movie...**

Press this button to begin producing a new movie. If you have already used the tool to produce a movie but have not yet saved it, you will be asked if you would like to save the movie before proceeding. Then, the dialog will be restored to its original default settings. Notice that this button is disabled until a movie has been recorded and is stored in memory.

### **Start Recording**

Movie Maker will begin capturing animation frames by launching the animation of the currently selected oneline diagram and storing each frame in a buffer in memory. If you have chosen to use a fixed number of frames as the criterion for stopping the recording, then the frame capturing will cease after the specified number of frames have been saved. Otherwise, you will have to press the **Pause recording** button. If you press the **Start recording** button after having pressed **Pause** recording button, the new captured frames will be appended to the list of frames already in memory.

### **Pause Recording**

Use this button to pause the recording process. If you have specified that the movie should include a set number of frames, you likely will not need to press this button. However, you may, in fact choose to use it. You can then press the **Start recording** button again, and the new screen frames will be appended to the list of frames already stored in memory.

### **Save Movie**

Press this button to store the sequence of frames to a file in AVI video format. Simply provide a name for the movie, and Movie Maker will transfer the images to the file for later playback.

### **Play Movie**

If you press this button, Simulator will launch your system's movie player to play the movie you just recorded. If you have not yet saved the frames to a movie file, you will first be asked to provide a name for the movie file in which to save the captured images. Movie Maker always plays movies from a file rather than from the sequence of frames stored in memory.

### **Show Options**

When you record a movie, the Movie Maker dialog assumes a much smaller footprint so that it blocks as little of the animated oneline as possible. This hides the panel that houses the aforementioned options. If you want to restore Movie Maker to its original size, press this button.

### **Close This Form**

As the name suggests, pressing this button will close the Movie Maker dialog. If you have not already saved the movie to a file, you will be asked to do so.

# Contingency Analysis

## Overview

### Contingency Analysis: An Introduction

Contingency analysis is a vitally important part of any power system analysis effort. Whether you are investigating the long-term effects on the transmission system of new generation facilities or projected growth in load, or you are considering whether to accept a transaction for next-hour's energy trade, it is extremely important that you analyze the system not only for its current topology but also for the system that results from any statistically likely contingency condition. Industry planning and operating criteria often refer to the n-1 rule, which holds that the system must operate in a stable and secure manner for the loss of any single transmission or generation outage. Therefore, it is very important to pay attention to the effects contingencies may have on the operation of your system.

PowerWorld Simulator is equipped with a set of tools for analyzing the effect of contingencies in an automatic fashion. The current edition of Simulator can process lists of contingencies involving

- The opening or closing of transmission lines and transformers
- The loss or recovery of a particular generator, load, or switched shunt
- The movement of generation, load, or switched shunt MWs or Mvars.
- The changing or setting of load, switched shunt, or generator MWs or Mvars
- The opening of all lines connected to a bus
- The opening or closing of all lines or transformers in an interface.

Simulator can be set to use a full Newton solution to analyze each contingency or use a dc load flow method. The full Newton approach is not as fast as dc load flow, but the results tend to be significantly more accurate, and also allow for gauging voltage/var effects.

This section describes how to use Simulator's contingency analysis tools.

### Contingency Analysis Dialog

Simulator's contingency analysis tools can be accessed only from Run Mode. Select **Options/Tools > Contingency Analysis** from the main menu in Run Mode. In response, Simulator will open the Contingency Analysis Dialog. This dialog has four tabs, only one of which is active and visible when no contingencies have been defined. When you access the contingency analysis features for the first time, no contingencies are defined, so only the contingencies tab is visible. To define contingencies for the case in memory, you may either load a pre-existing list from a file (see Loading Contingencies from Files), or you can use the Auto Insert Contingencies utility. Alternatively, you may insert contingencies one at a time into the topmost grid on the Contingencies tab by right-clicking on the grid and selecting **Insert** from the local menu to bring up the Contingency Case Dialog. Once you have defined contingencies for your case, all four tabs of the contingency analysis dialog will be enabled.

Each tab of the contingency analysis dialog covers a particular aspect of the analysis.

The Contingencies Tab is the only one of the tabs that is always visible. It 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.

Finally, 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.

With this overview in place, we can now take a closer look at how to analyze contingencies in Simulator.

### Contingency Case Dialog

The Contingency Case Dialog serves as an information source for each contingency that has been defined for the case. You can use the Contingency Case Dialog to scroll through the list of contingencies, to view and modify their definitions, and to insert new contingencies. You may access this dialog by choosing either **Show Dialog** or **Insert** from the local menu of the Contingency Records Display. The Contingency Case Dialog has the following controls:

#### Contingency Label

Identifies the name of the contingency currently being described by the dialog. Use the dropdown arrow to select a different contingency, or use the scroll buttons to navigate through the list of contingencies.

#### Add New Contingency

Prepares the dialog for the addition of a new contingency to the contingency list by clearing the Contingency Elements list and creating temporary storage for the new item. To add a new contingency using this dialog, first type an unused contingency name in the **Contingency Label** field, and then click **Add New Contingency**. The name you supply must be unique; otherwise, you will receive a warning indicating that the name you have specified is already used by a contingency in the list.

#### Insert New Element

Click this button to add a new action to the contingency. This will open the Contingency Element Dialog, which will allow you to define the new action. When you return to the Contingency Case Dialog, the Contingency Elements Dialog will contain the newly inserted action.

#### Clear All

Removes all actions from the contingency definition. The Contingency Elements Table will then appear blank, indicating that the contingency involves no associated actions.

#### Contingency Definition

This dialog contains a Contingency Definitions Display which lists the actions that comprise the contingency. To add actions to the contingency, right-click on the display and choose **Insert**. This will bring up the Contingency Element Dialog where you can define the action. To delete an action, right-click on the display and choose **Delete**. For more information about this display, see the Contingency Definitions Display.

#### Define Solution Options

Click this button to open the Contingency Solution Options Dialog. Use this dialog to define power flow solutions options for use under this contingency.

#### Use Specific Solution Options

Once you have defined solution options for this contingency, check this box to enable their use.

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 contingency from the contingency list.

## 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 Case 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 what the power flow solution options.

In order to specify options for a specific contingency, you will click on a button on the Contingency Case Dialog. In order to specify options for all contingencies, you will click on a 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 options, 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 as for Power Flow** button. To set all options back to use default, click the **Clear All Settings** button.

## Contingency Element Dialog

The Contingency Element Dialog provides information on the individual actions that comprise a contingency definition. You may use this dialog to modify an existing contingency’s definition, to add elements to an existing contingency or one that you are creating, and to delete an existing contingency. You may open this dialog in any of two ways:

- By pressing the **Insert New Element** button on the Contingency Case Dialog.
- By right-clicking on a Contingency Definition Display and choosing **Insert** or **Show Dialog**.

The Contingency Element Dialog features the following controls:

#### Element Type

Indicates whether the element involved in the contingency action is a branch, generator, load, switched shunt, bus, or interface.

### 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 following five options are available.

- Open: available for all element types
- Close: available for all element types except a bus
- Move: available for loads, generators, and switched shunts
- Set To: available for loads, generators, and switched shunts
- Change By: available for loads, generators, and switched shunt

Make-up Power Sources allows you to define other loads and/or generators to account for the change in supply or demand caused by performing a specified contingency action. The **Amount** field displays the amount of generation, load, or switched shunt amount specified for the contingency. You may also specify whether the amount being moved is in MW with a constant power factor, percent, MW, or Mvar.

### Comment

A user-specified comment string that can be associated with this action. While this comment is not used by Simulator in any way, the contingency auxiliary data files will save these comments 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 add a sentence explaining why the action is only performed under the specified criteria.

### Model Criteria

Click the Add button to specify a criteria under which this action will occur. 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.

Click **OK** to save any changes you have made to the contingency action, or click **Cancel** to close the dialog without saving your changes. Click **Delete** to remove the action from the contingency. Finally, click **Help** to view this help screen.

## Make-Up Power Sources

Generator and load contingencies may cause imbalances between generated power and demand. The default way to handle these imbalance is to assign them to the system slack. However, this is probably unrealistic and, depending on the size of the unit or load involved, it may cause you to have convergence problems.

To implement more local compensation for generation and load 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 or load 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.

Compensators account for the changes caused by a generator or load contingency by changing 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 doesn't 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.

## PSS/E Contingency Format

Simulator can read and write parts of the contingency format used by Power Technology's PSS/E Versions 23 through 27. The current version of Simulator does support 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.

## 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 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 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.

## Contingency Analysis Dialog Overview

### 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. For example, you can insert a new contingency record by selecting **Insert**, or you can delete a particular contingency by selecting **Delete**. You can display the contingency case dialog corresponding to a particular contingency by selecting **Show Dialog** from the local menu. You can copy portions of the display to other applications, print the contingency list or save it as an HTML table, or save the records to a file in a format that Simulator can read later. You can 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 will display the word **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.

**Violations**

Identifies the number of violations caused by the particular contingency. This number represents the total number of contingencies (branch thermal violations + bus voltage 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 Line %**

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 Definition Display

The contingency definition display identifies the actions that comprise the contingency selected. This display appears on the Contingency Tab of the Contingency Analysis Dialog, and also on the Contingency Case Dialog.

To add actions to the contingency, right-click on the display and choose **Insert**. This will bring up the Contingency Element Dialog where you can define the action. To delete an action, right-click on the display and choose **Delete**.

The contingency definition display is a type of Case Information Display and thus shares many characteristics and controls common to all other case information displays. The Contingency Definitions 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 either Name, Num, Name/Num, PW File Format, 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 Contingency Definition 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.

This feature allows you to enable or disable a particular contingency 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, the contingency auxiliary data files will save these comments 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.

## 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 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. By default, the Contingency Violations Display contains the following fields:

**Category**

The type of violation that occurred.

**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 Chester 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

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

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.

## Lines, Buses and Interfaces Tab

The Lines, Buses and Interfaces Tab is somewhat the opposite as the Contingencies Tab. This page allows the user to display all the buses, lines and interfaces in the case and see how many times a violation occurred on each 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.

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 move 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.

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.

This tab also 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. The 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.

## Other Contingency Actions

By clicking the **Other Actions** > button on the Contingency Tab or Line, 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 case presently set the present power system back to the contingency analysis reference state. For more information on the reference state, see Contingency Case References.

### 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.

### Compare Two Lists of Contingencies

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 Reports.

### 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 violation 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.

### Sensitivity Calculations >

Simulator provide 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 clicking on this. 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**. See the note above regarding the **Filter Results using Limit Monitoring Settings** to better understand how inactive violations are treated.

## Comparing Contingency Analysis Results

With previous versions of PowerWorld Simulator, users have been able to perform automated contingency analysis on a list of contingencies. One could then save this list and the corresponding results in a file for viewing at a later time. Starting in Simulator Version 7.0 you could now compare the results for two separate lists of contingencies. This functionality has been integrated into the existing Contingency Analysis Dialog. To compare two sets of contingencies, do the following:

- Process each set of contingencies and save the results for each list in a separate file.
- To save a contingency list, right click on the list of contingencies and choose **Save Contingency Analysis File...**
- Choose a name for the file, and the click **Save**.
- You will then be prompted with the question **Do you want to save the limit violations, too?**
- Click **Yes** so that the results of the analysis are saved with the list of contingencies.
- Now right click on the contingency list and choose **Compare Two Contingency List 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.

#### 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 that the Controlling Contingencies will be compared to. 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 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.ctg. 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.ctg.

You should define the contingency results you are more interested 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.ctg is set as the Controlling List and comparison.ctg as the Comparison List.

Now consider a specific contingency called CTG1. In comparison.ctg CTG1 caused an overload on LINE1. In controlling.ctg CTG1 did not any overloads at all. Therefore, on the dialog when you have selected CTG1, the Violations List will say **None Defined**. This is because the Violations List only shows violations in the Controlling List.

Note: if you want to see violations in file comparison.ctg which are not in controlling.ctg, simply define the file comparison.ctg is set as the Controlling List and controlling.ctg as the Comparison List.

- After clicking OK on the dialog, the contingency lists will be read from the specified files. 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 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 Line % (Max Interface %)

The highest branch (interface) violation caused by this contingency in the Controlling List.

#### Comp Max Line % (Comp Max Interface %)

The highest branch (interface) violation caused by this contingency in the Comparison List.

#### Worst Branch Increased Violation (Worst Interface Increased Violation)

The worst increase in a branch (interface) violation from the comparison list to the controlling list.

#### Worst Branch New Violation (Worst Interface New Violation)

The worst new branch (interface) violation

#### 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 Increased Violation (Worst HighV Increased Violation)**

The worst change in a violation.

**Worst LowV New Violation (Worst HighV New Violation)**

The worst new violation.

**Worst LowV Violation (Worst HighV Violation)**

The worst 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 relevant to this violation.

**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.

**Worst New Violation**

The worst new branch (interface) violation.

**Worst Violation**

This is the maximum of the following two values: [**Worst Increase Violation**] and [**Worst New Violation - 100%**]

**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.

**Worst Max Volt CTG New**

The worst new high voltage violation.

**Worst Max Volt CTG**

The worst 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.

**Worst Min Volt CTG New**

The worst new low voltage violation.

**Worst Min Volt CTG**

The worst low voltage violation.

## 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 is occupied by a four-page notebook, each page of which concerns a different aspect of the contingency analysis. These tabs are:

- Modeling
- Limit Monitoring
- 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, if you have never clicked **Set Options**, to the option settings that existed when the Contingency Analysis Dialog first opened), click **Reset**.

### Options Tab: Modeling

#### Calculation Method

The calculation method defines how the load flow is solved during the contingency analysis. By default, Simulator uses a Full AC Power Flow for each contingency. For a large set of contingencies and a large case, this can take some time to complete. Alternatively, you can now choose to solve the load flow using either Linearized Lossless DC or Linearized Lossless DC with Phase Shifters methods. These two methods will both solve a set of contingencies much faster than the full AC load 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 normal 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.

#### 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**, **Assuming constant voltage magnitude**, or **Assuming reactive power does not change**.

#### 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 Contingency Solution Options** button to open the Contingency Solution Options Dialog.

#### 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. You can specify to have each Model and Filter verified for the contingencies, or to assume that the Models and Conditions are either always true or always false.

#### Make-Up Power

The Make-Up Power grid defines how the system reacts in the case of a generator outage. In this display is a list of all defined areas in the system and a percent contribution amount. The value assigned to the contribution amount determines how much each area will make up for the outage. For example, if there are four areas and each is defined at 25% contribution, an outage in one area would be compensated by 25% extra from its own area and 25% from each of the other areas. This helps spread the strain out among the entire system instead of relying on the slack bus.

## Options Tab: Limit Monitoring

The Identify Violations Tab allows you to shape how limit violations are detected.

### 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 base case full AC load flow operating point just prior to the contingency calculations.

### 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 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.

## 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 what goes in that 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 display all of the other options. Although all of the other options will be activated, none of the other options will be checked.

### 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.

### 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.

### Create database-friendly tables

Checking the create database-friendly tables 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.

## Options Tab: Miscellaneous

### Automatically load the contingency file when you load the case

If this option is checked, then the file will be loaded whenever you load this case. This will work only if the contingency file and the case file are located in the same directory. This option is not available to you until either you load a contingency file from disk or save the current list of contingencies to disk.

### 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 a file when the list of contingencies is saved to a 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.

## Summary Tab

The Summary Tab of the Contingency Analysis Dialog provides additional information on the status of the contingency analysis run, 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**.

## Contingency Analysis Options

### Auto Insert Contingencies

Simulator allows you to generate automatically a contingency list containing all single branch and all single unit outages. To accomplish this, click the **Auto Insert** button on the Contingency Analysis Dialog. This opens the **Auto-Insertion of Contingencies Dialog**. This dialog is used to produce a contingency list automatically.

Press the Insert Contingencies button to generate the contingency list. Click Cancel to abort the automatic contingency creation.

It contains the following controls:

#### Automatically generate contingencies 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, or generating unit contingencies if you want each contingency to contain only one contingency 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 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.

#### 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 Filters dialog. You may edit the Area/Zone filters by clicking on the button as well.

### **Only include branches meeting an advanced filter**

When checked, the branches included in the auto-generated contingencies will be only branches meeting conditions outlined by an advanced filter. To set the conditions to be used, click the **Define Branch Filter** button.

### **Only include generators meeting an advanced filter**

When checked, the generators included in the auto-generated contingencies will be only generators meeting conditions outlined by an advanced filter. To set the conditions to be used, click the **Define Generator 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 bus of bus 1, then the contingency record will include the branch between buses 1 and 2, and if desired any generators attached to buses 1 and 2. However the branch between buses 2 and 3, and any devices attached to bus 3, will NOT be included in the contingency, because bus 3 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 three fields allow you to set a specific prefix for generators, lines, and transformers 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, and X for transformers. 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.

Press the Insert Contingencies button to generate the contingency list. Click Cancel to abort the automatic contingency creation.

## **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). This format is used in PSS/E versions 23 - 27.

To load contingencies from a text file, click the **Load** button on 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.

## Saving Contingency Records to a File

To save the contingency records you have defined for a case, either click the **Save to File** button on the contingency analysis dialog or select **Save Contingencies to File** from the local menu of the contingency records display.

Simulator can save contingency definitions to 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). This format is used in PSS/E versions 23 – 27.

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 formatted files

- Does not support any actions regarding an Interface

## Contingency Case References

Contingency analysis always stores a Reference State or pre-contingency state that is used throughout contingency analysis. Prior to solving a contingency, this reference state is always loaded into memory, so that all contingencies start from a common Base Case.

### Defining the Reference State

The reference state is initially defined as the power system state that exists at the moment that the first contingency record is defined for your system, either through loading contingencies from a file, creating them individually, or auto-inserting them. To change the reference at a

later time, you may choose the **Set Reference** option from the Other Actions > button on the Contingencies Tab of the Contingency Analysis Dialog.

During automatic processing of the contingency list, the reference state is always loaded into memory prior to the execution of each contingency, so that all contingencies start from a common Base Case. Furthermore, at the end of the automatic processing, the system is set back to the reference state.

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** will cause Simulator to first load the reference state into memory, and then execute the contingency. The system state will then reflect the power system flows of this post-contingency state. You may then modify the power system in any manner you wish, but realize that prior to solving another contingency, Simulator will reset the system state to reference state. The advantage of this approach is that you can implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by this contingency. These modifications will then be removed as soon as you move to the next contingency.

**Solve and Set As Reference** acts the same as Solve Selected Contingency with one exception. After executing the contingency, however the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use this as the Reference State.

## Fault Analysis

### Fault Analysis

Fault analysis can only be performed when Simulator is in Run Mode. There are four ways to start a fault analysis study:

- From the Options/Tools menu, select Fault Analysis...
- From the Run Mode Palette, click the **Fault** button
- 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 four 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 **Options/Tools** menu or the **Fault** button, 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, transmission lines, and transformers.

The layout of this dialog is described below:

#### 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:

- |                                  |  |
|----------------------------------|--|
| <b>Single Line - to - Ground</b> | Computes a single phase line - to - ground fault using a user defined ground fault impedance. The phase evaluated is always referenced as phase A. |
| <b>Line - to - Line</b>          | 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.    |
| <b>3 Phase Balanced</b>          | Balanced three-phase line fault - to - ground using a user defined ground fault impedance.   |
| <b>Double Line - to - Ground</b> | Computes a line - to - line - to - ground fault, using a user defined ground fault impedance.  |

#### XF Wye-Delta Phase Shift

The check box labeled **Include in Calculation** allows you to choose to have Simulator automatically calculate the phase angle references of the buses in the system using the transformer configurations. If checked, Simulator will search through the sequence data of all the transformers in the case, and automatically adjust the reference angle for each bus according to wye-delta and delta-wye transformer configurations. 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.

The wye-delta and delta-wye configurations do not override an existing phase shift entered in the Phase field for a transformer. In other words, if Simulator is automatically adjusting the angle references based on the transformer configurations, Simulator will combine the hard-coded angle (in the Phase field of the transformer tab on the transmission line dialog) with the 30 degree shift due to a delta-wye or wye-delta configuration.

Lastly, if the box is unchecked, then Simulator will set all buses in the system to a 0 degree reference, regardless of transformer configurations. Again, if a phase shift is hard-coded for a transformer, the shift is incorporated in the sequence admittance matrices for solving the fault, but the resulting fault voltages and currents are all given on a 0 degree reference.

#### Current Units

Allows you to choose to observe the fault currents in per unit current or actual Amps.

#### Data Type Shown on Online

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 oneline diagram.

The fields necessary on the oneline 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.

#### 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.

### 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

## 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 that 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, either click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting local menu, or choose **Options/Tools, Contouring** from the main menu.

## 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, either click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting local menu, or choose **Options/Tools, Contouring** from the main menu. 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 object; the options are Bus, Line, Interface, or Area/Zone. This selection narrows the set of quantities that can be contoured, which is specified in the **Value** dropdown box.

### Value

Then select the quantity to contour from the Value dropdown box. See NOTE at the end of this help topic.

### Pixel Options

In order to optimize the speed of drawing, the user must specify the Influence Region as well as the Contour Resolution. Also, when contouring line or interface objects, the number of data points used to represent that line must be chosen.

### No Data Color

This setting allows you to choose the contour color around devices that have no 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.

### Save Contour Image with Oneline

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.

### 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.

### Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values.

### 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.

### Brightness

Modify the brightness track bar to change the brightness of the color map.

### 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.

### Values

These values along with the color map define how to convert your values into a color for the contour. The values are:

<b>Absolute Minimum</b>	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.
<b>Limit Minimum</b>	This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.
<b>Nominal</b>	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.
<b>Limit Maximum</b>	This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.
<b>Absolute Maximum</b>	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.

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.

### 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 MVR
- 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 object; the options are Bus, Line, Interface, or Area/Zone. This selection narrows the set of quantities that can be contoured, which is specified in the **Value** dropdown box.

### Value

Then select the quantity to contour from the Value dropdown box. Note: see Contour Type for more information.

### 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.

### 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.

### 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 weighted average 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 to do a weighted sum method will correct the disparity, and the contour around these two different groups of generation would look basically the same.

### Exaggerate Limit Violations by

To cause even minor limit violations (values outside of 25-75%) to stand out prominently on your contour, check the Exaggerate Limit Violations by check-box. Then adjust the sliding scale to modify how much the violations will stand out.

### 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.

### Fade to Value

The value to which a data point's value fades as it moves away from its location.

## 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.

### Brightness

Modify the brightness track bar to change the brightness of the color map.

### Contour Type Values to Use

These check-boxes signify which values from the Contour Type Values Tab are used by the Color Map. The Nominal Value must always be used and therefore cannot be unchecked.

### 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.

### Save As New

To save the present color map as a new color map, click this button. Then specify a name for the new color map.

### Save Map

To save changes that have been made to the present color map, click this button.

### Rename Map

To rename the present color map, click this button.

### Delete Map

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.

**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.

## Distribution Factors

### 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 different areas or zones. 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). 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.

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 the output of its generators by 100% of the transfer amount, while the Buyer decreases the output of its generators by 100% **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 decrease in the Buyer's generation will be greater than 100% of the transfer.

#### To Calculate the Power Transfer Distribution Factors:

- Perform an initial Power Flow Solution.
- In Run Mode, select **Options/Tools, Power Transfer Distribution Factors (PTDFs)** from the main menu 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, 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. You access this dialog by selecting **Options/Tools, Sensitivities, Power Transfer Distribution Factors (PTDFs)** from the main menu in Run Mode only. The dialog has the following options:

#### **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 PTDF's.

**Seller, Buyer**

These dropdown 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 PTDF's.

**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 PTDF's.

**PTDF Type**

This option is only available if you have the Optimal Power Flow (OPF) add-on for PowerWorld Simulator. 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 define multiple directions, the **Seller** and **Buyer Type** fields are replaced by the Direction Records display for viewing and defining directions.

**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.

**Calculate PTDFs**

Click on this button to update the PTDF values. The results table will reveal the latest calculations.

**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.

**Total 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.

**Automatically Update**

If checked, the PTDFs are automatically updated every time the power flow is solved.

**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 filter set.

**Only Show Values Above %**

Restricts the result set to show only those PTDFs that exceed a specified value. Many line-loading relief routines, for example, disregard elements having a PTDF of less than 5%.

**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.

**Highlight Counter Flows**

If this box is checked, then the PTDFs that are directed counter to the existing flows on the system are highlighted on the oneline using the *Counter Flow Highlight Color*. Double click on the color to change its value.

**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 Bus #, From Bus Name, To Bus #, To Bus Name, Circuit**

Identifiers for the transmission line or transformer.

**DF From Bus %**

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.

**DF To Bus %**

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.

**Base KV (Max) and Base KV (Min)**

Displays the maximum and minimum base voltages for the line. This is useful for identifying transformers and which end of the PTDF relates to which base voltage.

**Interfaces**

Shows the transaction distribution factors for the interface records. The following fields are shown:

**Interface Name**

Alphanumeric identifier for the interface.

**Interface Number**

Numeric identifier for the interface.

**Dist. Factor %**

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.

**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:

**Bus Number, Bus Name, Gen ID**

Generator's terminal bus number and alphanumeric identifier, and the id for the generator.

**Area Number, Area Name**

Name and number of the generator's area.

**Gen Change %**

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.

## 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.

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 show up on other transmission line after the insertion.

**To calculate the LODFs:**

- Perform an initial Power Flow Solution.
- In Run Mode, select **Options/Tools, Other Sensitivities, Line Outage Distribution Factors (LODFs)** from the main menu 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

$$OTDF_x = PTDF_x + LODF_{x,y} * PTDF_y$$

where  $PTDF_x$  and  $PTDF_y$  are the PTDFs for line x and y respectively, and  $LODF_{x,y}$  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

$$OutageFlowX = PreOutageFlowX + LODF_{x,y} * PreOutageFlowY$$

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 **Options/Tools, Other Sensitivities, Line Outage Distribution Factors (LODFs)** from the main menu in Run Mode only. The dialog has the following options:

**From Bus Number, To Bus Number, Circuit**

Specify the line whose status modification you would like to determine sensitivities to.

**Action**

Check Outage Sensitivities to determine sensitivities for the outage of a line.

Check Insertion Sensitivities to determine sensitivities for the insertion of a presently outaged line.

**Liner Calculation Method**

**Linearized AC**

This calculation method is not yet available.

**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.

**Calculate LODFs**

Click this to calculate the LODFs and update the display.

**LODFs Tab**

This tab contains a table showing a list of the lines in the case. Since this table is 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

- This dialog allows you to calculate the LODFs for several different contingent lines. Go to 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.
- 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.

### 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 Dialog

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 Type, Sink Name

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.

## Directions Display

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 PTDF's.

**Processed**

Displays if the direction has already been processed for a multiple direction PTDF analysis.

## 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. The layout of the dialog is as follows:

**Type of Direction**

There are four types of directions that can be automatically defined. **Area to Slack** and **Zone to Slack** will define directions from areas or zones to the slack bus of the system. **Area to Area** and **Zone to Zone** will define directions from areas to other areas or zones to other zones.

**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 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.

## Sensitivities

### Flow and Voltage Sensitivities

The Flows and Voltages Sensitivities Dialog can be opened from the **Options/Tools, Sensitivities, Flows and Voltages** menu option in Run Mode.

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. The grid that occupies the bottom of the dialog lists each bus in the system, subject to the Area/Zone 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, MVR, or MVA flow, as dictated by the **Flow Type** setting) on the device identified by the **Device Identifier**. Likewise, the *Q Sensitivity* field indicates the effect a 1 MVR increase in reactive power at the bus has on the flow (either MW, MVR, or MVA flow, as dictated by the **Flow Type** setting) on the device identified by the device identifier.

Use the **Device Type** control to indicate whether the sensitivities are to be calculated for a line/transformer or an interface. Use the **Flow Type** control to specify the type of power flow for which the sensitivities will be calculated. Use the **Device Identifier** fields to identify the line/transformer or interface.

Whenever you make a change to any of these settings, click **Calculate Sensitivities** to update the grid with the new sensitivities.

Click **Close** to close the Flows and Voltages Dialog.

### Loss Sensitivities

The **Losses Dialog** can be opened from the **Options/Tools, Sensitivities** menu option.

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. The display is available in the Run Mode by selecting **Options/Tools, Sensitivities, Losses**.

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 five values:

#### None

No Losses are calculated because a loss function is not specified.

#### Each 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.

**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.

**Selected Areas**

Losses are calculated with respect to the total losses for a group of areas, specified in the Selected Areas field.

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 Bus Marginal Loss Sensitivities Dialog houses the following controls:

**Loss Function Type**

Use this to specify which Loss Function to use.

**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 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:

<b>Number, Name</b>	Number and name of the bus.
<b>Area Number, Area Name</b>	Number and name of the bus' area.
<b>MW Sens.</b>	Sensitivity of the loss function to an increase in the real power injection (generated power assumed positive) at the bus.
<b>MVR Sens.</b>	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 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 may be thought of as the inverse of 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 **Options/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. The following describes the sections of the dialog:

### Device Type

Select whether you want to calculate the sensitivities for a transmission line/transformer or for an interface. To calculate sensitivities for an interface, you must have the interface defined in the case.

### Device Identifier

Specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name, number, and monitored flow direction.

### 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, and Participation Group.

### TLR Sensitivities

Specify if the next set of calculated TLR sensitivities should replace the currently calculated values, or be appended to the current values.

### PTDF Calculation Method

Choose the solution method to use for calculating the sensitivities.

**Include only AGCable Generators**

If checked, then only generators available for generation control will be included in the TLR sensitivity calculations.

**Generator, Area and Bus Sensitivities**

Displays the other devices that can be involved in a transaction with the transactor, and the resulting TLR sensitivity for performing a transaction with that device.

## 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)
- Available Transfer Capability Analysis (ATC)

### Voltage Adequacy and Stability Tool (PVQV)

#### PowerWorld Simulator PVQV Overview

**Note: The PVQV option in PowerWorld Simulator is only available if you have purchased the PVQV add-on to the base package. To learn more about the PVQV, please feel free to read through the information contained in these help files. Contact PowerWorld Corporation for details about ordering the PVQV version of Simulator if you do not presently own it.**

The purpose of this portion of the help is to teach you how to use PowerWorld's voltage stability assessment tool, **PVQV**, to analyze the voltage characteristics of your system. Since this application is currently under continuous development, bear in mind that certain details described here may change and thus render portions of this document inaccurate. However, PowerWorld Corporation will try to clarify the implications of significant changes as they arise. If you have any questions concerning the use of the product, suggestions on how to improve it, or information concerning program bugs that you have observed, please contact PowerWorld Corporation at [info@powerworld.com](mailto:info@powerworld.com).

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 tool (**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. PowerWorld Corporation also has optimal power flow (OPF) and available transfer capability (ATC) add-ons to the base package.

All of the PVQV commands and options are accessed using the Voltage Stability 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 to allow the user to monitor any system parameter while automatically increasing a user-defined transfer. The PVQV module uses the Simulator built-in Newton Raphson power flow algorithm to accomplish this task. After the PVQV simulation is completed, the user can choose to graph any of the monitored system parameters.

The most efficient way to learn how to use this tool is to do so by example. Thus, the help topics in this section will provide step-by-step instructions on using the voltage stability assessment tool to analyze a case. It is recommended that you pick a case relevant to your own use and go through the help topics step by step with that case.

## PVQV: Obtain an Initial Solution

Load a case into Simulator by choosing **File / Open Case** from the main menu and selecting the case you wish to load. You are now ready to work with the case. Simulator has two modes of operation: *Edit* and *Run*. Edit Mode is used to edit a case, while Run Mode is used to analyze and simulate it. Switch to Run Mode now. Solve the case by pressing the button labeled **Single Solution** near the top of the screen, just below the main menu. If this button is not visible, choose **Simulation / Single Solution - Full Newton** from the main menu to perform the solution, or select **Windows / Toolbars / Program** from the main menu to expose the button. If the case is solvable, the solution will be reached in a few iterations. You can keep track of Simulator's progress in solving a case by watching the Message Log. If it is not already visible, expose it by clicking the button labeled **Log**. (Again, if this button is not visible, expose it by clicking **Windows / Toolbars / Program**.)

If Simulator fails to solve the case, you must rectify the problem before proceeding. You may try to enforce MVAR limits immediately to get it to solve, or you may need to disable some automatic controls. More details on how to perform these tasks can be obtained in other help topics. However, if the case you're working with was previously solved, you shouldn't have any problems solving it in Simulator.

## PVQV: Consider What to Do

The voltage stability analysis process involves increasing a transfer of MW from one region of the system to another and monitoring what happens to system voltages as a result. This type of analysis is commonly referred to as a PV study. Before you can proceed, then, you obviously need to define which region or regions will import the transfer, and which region or regions will supply it. Simulator provides objects called Injection Groups to allow you to define groups of generators or loads that will act in unison to implement a power shift.

You may need to analyze the stability of the system not only for its current topology but also in the event of specific contingencies. Simulator has a contingency analysis feature that analyzes the effects of topology changes using full AC analysis. It has been integrated into the voltage stability analysis to allow you to gauge the impact of contingencies on your system's stability margins.

Before proceeding, you need to define the injection groups that will represent your source and sink for the PV study. Also, if you want to consider contingencies, you will need to develop the contingency list.

## PVQV: Defining Injection Groups

For this example, we will study a transfer of power from generators in one area to an increasing load in another area. We need to define an injection group containing the generators that will serve as the source, and another containing the loads that will serve as the sink. Here is the procedure:

- Select **Case Information / Area/Zone Filters** from the main menu
- One of the columns of the display is labeled '**Shown**' and its entries have values of either '**YES**' or '**NO**'. Right-click in this column and select '**Toggle all No**' from the resulting menu.
- We will first define the source areas. Scroll down the **Area / Zone Filters** display until you find the record for source area. Double-click its '**Shown**' setting, which is currently '**NO**', so that it becomes '**YES**'. Do the same for the any other areas you would like to have sources.
- Choose **Case Information / Injection Groups** from the main menu. This opens the Injection Groups Display. Since there are no injection groups defined yet, the display has

a single entry that reads 'None Defined'.

- Right-click on the Injection Group Display and select **Insert** from the resulting menu. Type a name for the injection group in the resulting dialog. This will begin the process of creating a new injection group that will consist of the generators in the areas you selected.
- The Injection Groups Dialog will pop up. In the grid that occupies the right half of this display, click the right mouse button and select **Insert**. This opens the Add Participation Points Display. We use this form to define the generators that make up the injection group.
- Select all the generators listed in the box. You can easily select all the generators by first selecting the top one, scrolling down to the bottom of the list, and clicking the last one while holding down the shift key.
- For this example, we will assume that all source generators will participate in the transaction proportionally to their MW reserves. Therefore, select the **Base on Maximum - Output** option.
- Click the arrow button. This will add all the selected generators and their newly calculated participation factors to the list box on the right.
- Click **OK**. The Injection Groups Dialog will identify the participation points we've just added for injection group.
- Click **Close**.

We now must define the sink group. The procedure is very similar:

- Open the Area/Zone Filters Display and toggle the '**Shown**' values so that only areas that you want to use as load sinks have a value of '**YES**'.
- Open the Injection Groups Display. Right-click it and select **Insert** from the resulting menu. Supply a different name at the prompt.
- The Injection Groups Dialog will open. In the grid that appears on the right side, click the right mouse button and select **Insert** from the resulting menu. This opens the Add Participation Points Dialog. We want to have loads serve as the injection points for this group, so switch to the tab labeled '**LOADS**'.
- Select all entries from the list of load points, and choose the '**Base on Size**' option so that larger loads participate more heavily.
- Click the arrow button. This will add all the selected loads and their newly calculated participation factors to the list box at the right.
- Click **OK**. The Injection Groups Dialog now lists the participation points we've just added for injection group.
- Click **Close**.

We have now defined a set of source points and a set of sink points. The PV study will model an increasing transfer of power from these source points to the sink points.

Although this procedure seems to involve a lot of steps, it can be accomplished in about a minute once you get used to it. Moreover, there is a way to save injection groups so that they can be used with other cases. To save the two injection groups you just defined, follow these steps:

- Select File / Save/Load Auxiliary Data from the main menu.
- Click the button labeled **Save Injection Groups** and select a filename.

To load participation groups from a file,

- Select File / Save/Load Auxiliary Data from the main menu.
- Click the button labeled **Load Auxiliary Data**, and select the file containing the injection

group definitions.

If you know that there are specific sets of sources and sinks that you will want to study, it would be a good idea to make up a file containing these injection group definitions. Then, whenever you read in a new case, you can load these definitions from the file using the above procedure.

## PVQV: Defining Contingencies

Simulator can recognize and process contingencies expressed either in PowerWorld's or PTI's contingency formats. If you have a file of contingencies you wish to process as part of the voltage stability run, make sure that it is accessible from the computer you are using to perform the analysis. Simulator also provides a tool for defining and processing contingencies. This can be accessed by selecting **Tools / Contingency Analysis** from the main menu.

As an example, consider the four-element contingency WPS-ARP2e, which is a common contingency for the MAIN region in the central United States. Here we assume that this single contingency is expressed in PTI format and is stored in file ARPINCTG.CTG. ARPINCTG.CTG thus contains the following text:

```
CONTINGENCY WPS-ARP2e
OPEN LINE FROM BUS 39244 TO BUS 61853 CKT 1      /* ARP 345 345  EAU CL 3 345
OPEN LINE FROM BUS 61866 TO BUS 39706 CKT 1      /* T-CRNR7 115  WIEN  115
OPEN LINE FROM BUS 38342 TO BUS 39901 CKT 1      /* COC 69 69.0  COC DPC
OPEN LINE FROM BUS 38333 TO BUS 66027 CKT 1      /* HLT 69 69.0  MAUSTON 69.0
```

END

Of course, the contingency file can contain as many contingencies as you want to define. Because tracing a PV curve can be fairly computationally intensive, it is wise to limit the list to only the most critical contingencies, as identified by a contingency screener.

## PVQV: Define the Regions to Monitor

We've already used the Area/Zone Filters when we defined the source and sink injection groups. Simulator uses these filters to restrict the content of information displays to elements located in specific areas and zones. In particular, when monitoring system voltage violations, it is likely that you will want to restrict your focus to a certain few areas. To do this,

- Open the Area/Zone Filters display by selecting **Case Information / Area/Zone Filters**.
- Find the column labeled 'Shown'. Click the right mouse button in this column, and select **Toggle All No** from the resulting menu.
- Double click the 'Shown' value for a particular area. This changes its Display Filters value to 'YES'.

Repeat this last step for each of the other areas you would like to monitor.

The result is that all subsequent information displays will display information only for elements located within these areas.

## PVQV: Open the PV Study Form

If you have defined the source and sink injection groups, contingencies you'd like to study, and areas of interest, you are then ready to proceed with the study. To begin, select **Voltage Stability / PV Curves** from the main menu. This opens the **Power - Voltage Relationships** form. The form is organized in a series of tabs that are arranged in the order they should be considered. We

will use this form to set up an analysis of a transfer from source generation group to the sink load group.

## PVQV: How Should the System Vary

The first tab of the form, labeled '**How should the system vary?**', allows you to define the transfer to study in terms of its source, its sink, and how it should vary as the analysis evolves.

Click the arrow next to the **Source** box and select one of your source injection groups from the list. Click the arrow next to the **Sink** box and select your sink injection group from the list. This tells Simulator to study a transfer of power from the source injection group to the sink injection group. (If you had not already defined these injection groups, you could do so by clicking the **View / Define Groups** button and following the procedure detailed in the Defining Injection Groups section of this document.)

Recall that, when we defined the injection groups, we indicated that the contribution of each generator point should be proportional to its MW reserve. This is certainly a reasonable thing to do, but there may be occasions when you want to specify relative participation by area, not by individual buses. Specifically, assume that additional imports are to be supplied 10% by Area 1, 10% by Area 2, 30% by Area 3, 30% by Area 4, and 20% by Area 5. To adjust the contributions of individual areas in the source injection group in Simulator, click the button labeled **Areas** that is adjacent to the **Source** dropdown box. When you do this, a table appears showing the present percent contributions of the areas participating in the source injection group. To adjust the individual percent contributions, specify the new value for each area in the rightmost column, which is labeled **New % Contrib**. When you have finished, click the **Change Contributions By Area** button. The individual areas will then contribute according to your new specifications. If you want to do the same for the sink areas, click the **Areas** button that is adjacent to the **Sink** dropdown box, and follow the same procedure. In this way, you can designate which areas are supposed to participate most heavily in implementing the power transfer.

## PVQV: Options to Control the Transfer Increase

### Initial Step Size (MW)

This option indicates the initial rate at which the transfer will be increased with each successful iteration. For this example, set its value to 100. This tells Simulator to begin studying the transaction in 100 MW increments.

### Minimum Step Size (MW)

Whenever Simulator fails to solve the system at a given transfer level, it will return to the previously solved transfer level, reduce the step size by 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, Simulator 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. For this example, set the **Minimum Step Size** to 10.

### When convergence fails, reduce step by a factor of...

Whenever Simulator fails to solve the system at a given transfer level, it will reduce the transfer step size by the value specified for this option. For this example, set this value to 2. Therefore, Simulator 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, since it will conclude that it has arrived at the voltage collapse point, within the specified tolerance.

### Stop when transfer exceeds

Provide a MW transfer limit between the source and the sink. When the PV analysis reaches a transfer amount equal to this value, the PV analysis will terminate.

### Power Factor for Changing Load

This option controls how reactive load should vary as real load is changed during the analysis. You can specify a particular power factor at which the load should vary, or you can check the **Keep load power factor constant** box to force Simulator to increase real and reactive loads such that the power factor of each load remains constant. You will usually want to check this option.

### Allow only AGC units to vary

Control areas in Simulator may practice one of three types of automatic generation control. Then, all generators in the area that are, in fact, AGC-able will participate in the area's automatic generation control program. Thus, Simulator distinguishes individual generating units according to whether they do or do not participate in their area's AGC effort. By checking this option, you instruct Simulator to allow only those generators that are eligible to participate in AGC to contribute to the power transfer being studied. For this example, we will assume that all of the units listed in the source injection group, regardless of their AGC status, are to be used in providing power for the transfer. Thus, we will leave this option unchecked.

### 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. For this example, leave this option unchecked. This will allow us to analyze the capacity of the interface to support the transfer, regardless of the amount of reserves available.

### Do not allow negative loads

This is the analog of the previous option for loads. If a load is used as a **source** point, it will be decreased to make power available for the transfer. Checking this option will instruct Simulator 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. If you aren't using loads as source points this option is irrelevant. Thus, it can be left unchecked.

## PVQV: Defining What to Monitor

We will now define what quantities we want to monitor as the transfer is increased. Unmonitored system parameters will not be saved. Switch to the second tab of the form, which is labeled '**What should be monitored?**'. This page actually contains several sub-pages that allow you to monitor different types of objects, including buses, generators, injection groups, branches, interfaces, and system-wide quantities.

### Buses

For buses, we can monitor voltages, angles, MW load, and MVR load (an additional quantity, energy measures, is an experimental parameter and should never be selected for your studies). Initially, all values in this table equal '**NO**', indicating that we're not monitoring any bus-related quantities. To monitor a particular quantity, double-click the corresponding value to toggle it to '**YES**'.

The display makes it easy to find buses. Right-click anywhere on the display, and select **Find** from the resulting menu. Type the name of the bus for which you're looking, and click **OK**. Or, type the first few letters of the bus name for which you're looking, and click on the down arrow to show all the buses that start with those letters. Once you select a bus, the display will scroll to the bus you are seeking. Then, double-click that record's **Voltage?** value to change it to **'YES'**.

#### Generators

For generators, we can monitor MW and MVAR output, and MVAR reserve. The display operates exactly like the bus display.

#### Groups

The **Groups** sub-page allows you to monitor the MVAR reserves for a group of buses. This can be helpful, for example, when you have identified a group of buses that are instrumental in supporting voltages in a particular region, and you want to keep tabs on how much reactive reserve they have collectively as the transfer increases.

#### Lines / Transformers

The **Lines / Transformers** sub-page allows you to monitor real, reactive, and MVA flow on any branch in either the from-to or to-from directions. 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'**. For example, we could monitor the MVA flow from Eau Claire to Arpin, which are two buses in Wisconsin. To do this, right-click on the table and select **Find**. Then, enter the appropriate bus names or numbers and click **OK**. The display will scroll to the record corresponding to Arpin to Eau Claire. The line happens to be defined from Arpin to Eau Claire, so we will actually want to monitor flow in the to-from direction. Double-click the branch's **BRANCH MVA?** value twice to change it to **'TO-FROM'**.

#### Interfaces

Simulator allows you to define groups of branches that together comprise an interface. The **Interfaces** sub-page allows you to monitor real, reactive, and MVA flow on any interface that you have defined. If you have not defined any interfaces, you cannot use this option.

#### System

The **System** sub-page provides access to options that apply to the case in general. The following options are available:

##### Contingencies

To monitor the contingencies stored in a file, click the **Model contingencies contained in file** checkbox, and enter the name of the file containing the contingencies you want to monitor. You can click the **Browse** button to look for the file.

##### Voltage violations

Simulator needs to know whether to report buses whose voltage falls below a prescribed minimum voltage level or above a certain maximum voltage level. To tell Simulator to flag buses that have voltages below their allowed minimum, check the **"Low Voltage Violations** box. To flag buses that have voltages above their allowed maximum, check the **"High Voltage Violations** box. To set the minimum and maximum allowed voltages for each bus, click the **"Limit Group Definitions...** button to open the Limit Monitoring Settings Dialog. This will allow you to assign individual elements to monitoring groups that control the values for flagging violations for each element.

##### Inadequate voltage level

In performing these studies, we are usually concerned not only with voltage collapse, but also with inadequate voltages. We can define a particular voltage level to be inadequate. This voltage can be different from the level specified for the **Identify bus voltages whose pu value is...** option described previously. Note that you can also force Simulator

to terminate the run once it has detected an inadequate bus voltage by checking the box labeled **Stop when voltage becomes inadequate**. This is useful if you are interested in finding only the transfer level at which bus voltages fall below a certain level, and the collapse point isn't of interest.

## PVQV: Setting Output Options

Simulator records the value of each monitored quantity at each transfer level. However, unless you tell Simulator where to write the data, it will be present only in memory. The **Output Tab** allows you to designate where the data should be logged.

Click the **Save Results to File** option to indicate that you want to write results to a file. Then, in the adjacent text box, supply the complete path for the output file. You may use the **Browse** button to locate the place where you want the file to be written.

In addition to recording how the monitored quantities vary with the transfer, you can instruct Simulator to save the entire system state at regular intervals during the analysis. This can be helpful if you want to analyze particular transfer levels more closely after the analysis is complete. To archive system states, click the checkbox labeled '**Save system data every**', and specify the frequency with which the states should be saved. For example, if you want the system state to be saved at every iteration, specify a value of 1. If you want the system state to be saved every other iteration, specify a value of 2, and so on. Then, supply the directory where the states should be written, and a prefix to use in naming the state files. You can use this prefix to distinguish the states of different runs that might have been written to the same directory. If you do decide to use this option, keep in mind that, depending on the size of your system, archiving states frequently can require significant disk space and delay the process. However, it can be quite helpful if previous analyses have shown interesting behavior at a particular transfer level.

## PVQV: Other Options

The **Other Options** page offers additional choices for tailoring the analysis to your needs.

Use the **Margins** options to specify a cushion to maintain in reporting the voltage collapse point and the inadequate voltage point. Option **Collapse Cushion** specifies the percentage reduction to apply to the size of the transfer that results in voltage collapse. For example, if voltage collapse occurs for a transfer of size 1000 MW, applying a 10% voltage collapse cushion would mean that the transfer size that causes voltage collapse would actually be reported as 900 MW. Likewise, option **Inadequate Voltage Cushion** specifies the percentage reduction to apply to the size of the transfer that causes a bus voltage to become inadequate. For example, if an inadequate voltage were first found for a transfer of size 700 MW, a 10% inadequate voltage cushion would mean that the transfer size that causes inadequate voltage would actually be reported as 630 MW. Using these margins can help compensate for inaccuracies in the data and analysis techniques.

Click the **Solution Options** button to display the available power flow solution options. The power flow solution options provide control over the convergence tolerance, the behavior of automatic controls, and the enforcement of area interchange constraints. Information on setting these options is available by consulting the help files for Solution/Environment Options. You will likely not need to adjust these options for the voltage stability process.

Finally, you can force Simulator to pause between transfer levels by checking the **Pause Between Iterations** option. If this choice is selected, the voltage stability process will enter its paused state every time it computes the solution for a transfer level. This gives you time to analyze the resulting state more closely, or to save the state to disk.

## PVQV: Save Options for Future Use

The **Power-Voltage Relationships** form provides a great many options for customizing the analysis. However, you don't have to set all these options every time you perform the analysis. By clicking the button labeled **Save Options** at the bottom of the form, you can save the choices you've just made to a file. The next time you perform the analysis, rather than re-entering all these choices, you can simply click **Load Options** and specify the name of this file. All your choices will then be re-installed. This file will be saved as a text file.

## PVQV: Performing the Analysis

To start the PV study, switch to the **Control / Results** tab, the last page of the form. From this tab, you can control the progress of the run by initiating, pausing, or aborting it. You can save and load archived system states. You can even reset the process to the beginning. This tab also gives you a few different views of the run's output, including the ability to plot various quantities.

To begin the PV analysis, click **Run**. The **Status** indicator will change from **'Inactive'** to **'Running'**. To pause the process at any time, click the **Pause** button. Then, to restart the analysis, click **Run**. To terminate the analysis at any time, click **Abort**.

As the analysis continues, the **Summary** sub-page will keep you informed of its progress. The **Contingency** textbox will identify the contingency that is currently being modeled. **Step Size** identifies the current size of the increment the application is using to increase the transfer. **Total Shift** indicates the size of the transfer that was most recently solved. When the analysis has completed, **Collapse Margin** will indicate the size of the transfer that resulted in voltage collapse, modified by the **Collapse Cushion** you specified. When the analysis detects an inadequate voltage, **Inadequate Voltage Margin** will identify the transfer level at which inadequate voltages were first detected, modified by the **Inadequate Voltage Cushion** you specified. You will find that the **Summary** sub-page provides a concise overview of the study's key findings for the case currently being analyzed (whether that case is the pre-contingency model or one of the contingency scenarios).

The **Violations** sub-page identifies the bus voltage violations that are detected as the transfer increases. A table listing the bus voltage violations occupies most of this page. This table is automatically updated every time a transfer level has been solved. The bottom of the **Violations** display identifies the bus where inadequate voltage was first detected. The results reported on the **Violations** display depend on the rules you specified for identifying low, high, and inadequate voltage levels.

The **Plot** sub-page gives you access to the application's data plotting functions. Any of the values you designated to monitor can be plotted using this display

To produce a plot, follow this example procedure:

- From the X Axis Quantity dropdown box, select 'Total Shift (MW)'.
- For the X Axis Caption, type 'Total Transfer in MW'.
- From the Y Axis Quantity list box, select the first four monitored bus voltages.
- For the Y Axis Caption, type 'Bus Voltage (pu)'.
- For the Title, type 'Voltage vs. MW Transfer for SOURCE Selling to SINK'.
- To complete the procedure, click the button labeled Plot.

The plot will appear in its own window. You can produce as many plots as the memory on your system will allow. Right-clicking on a plot will expose a local menu with four options. You can send the plot to your printer by clicking **Print**. You can change the printer configuration by selecting **Printer Setup**. You can save the plot as a bitmap, Windows metafile, JPEG, or text file

by clicking **Save As** and selecting the appropriate file type. Finally, to close the plot window, click **Close**.

The final sub-page of the **Control / Results** tab is labeled **Contingencies**. This display summarizes the collapse margin and inadequacy margins obtained for the base case and all contingencies that have been studied.

## PVQV: Load State, Pause, Restore Original

There are four buttons in the left corner of the form. The buttons **Load State** and **Save State** enable you to load and saved archived system state information to / from disk. If you click **Load State**, then the model currently in memory will be updated to match the information stored in the state archive you are loading.

You may find that you would like to adjust the system model during the analysis, perhaps to test an operating procedure or automatic control scheme. To accomplish this, pause the simulation by pressing **Pause**, adjust the system to your liking, and click **Set Reference**. You may then resume the analysis with your new changes by clicking **Run**.

Finally, to restore the original case, click **Restore Original**. This re-creates the model as it was before the transfer was put in place. This essentially allows you to restart the analysis.

## PVQV: The Output File

Recall that you can designate a file as the place to log the results of the analysis. Note that this file is not intended to be a nicely formatted report. Rather, it simply lists for each monitored quantity the value that quantity attains at each transfer level. Each list is tab-delimited and can be easily imported into Excel for further analysis. Results for the pre-contingency case are saved to one file, while results for any contingency cases are automatically stored to separate, automatically named files. Results from contingency cases are stored in files having the same name as the output file you specified, but with the name of the contingency added as a prefix.

## PVQV: Q-V Curves

The other voltage stability function currently included is the ability to compute QV curves for any bus in the system. Simply select **Voltage Stability - QV Curves** from the main menu to open the QV Curve dialog. The QV Curve dialog allows you to specify the buses to be monitored, set defaults for the QV curves, and run the QV analysis. These three topics are handled on the following pages of the QV Curve dialog:

Buses

Options

Control/Results

When you are finished, simply click **Close**. The results will remain in memory as long as the case remains in memory. Please note that QV curve results and option settings are not saved with the case.

## PVQV: Q-V Curve Buses

To designate the buses for which you would like to calculate QV curves, toggle the **Selected?** field to **YES**. If you would like to specify specific solution parameters (min volt, max volt, and step size) for any bus, simply type the numbers in the appropriate cells. If you leave any cells blank, default values will be used for those parameters. Only non-default values will be shown in

these cells. If you change a value in these cells to a default setting, the cell will be automatically be set blank.

Alternatively, you may type a range of bus numbers to add them to the list of buses that will have a QV curve calculated, or you may type the name of a bus. Furthermore, you may load a list of buses, as well as other option settings, from an auxiliary data file. To create such a file to store your option settings, press the **Save Settings** button that appears near the bottom of the display.

To load settings you saved previously in an auxiliary data file, provide the full path in the text field that appears between the **Browse** and **Load** buttons. You can use the **Browse** button to search for the file instead of typing its name. Once the name of the file appears in the text field, press **Load** to import the settings stored in the file.

## PVQV: Q-V Curve Options

The second page of the QV Curve dialog is the **Options** page. The options page contains three tabs labeled **Solution**, **Output** and **Contingencies**.

### Solution

In the three text fields enclosed in the **Default solution parameters** group box, specify the values of voltage step size, minimum voltage, and maximum voltage to use when computing QV curves for buses that don't have specific values set for these parameters.

If you don't understand the meaning of these parameters, consider how a QV curve is computed: a fictitious generator (i.e. voltage source) is placed at the monitored bus. Its set point voltage is varied in steps of the specified size between the specified maximum and minimum voltages, and the MVAR injection at the bus is calculated and recorded at each step.

To make the changes you've made register with QVCC, click the **Set Options** button. To restore the factory-default settings, click **Restore Defaults**.

To set power flow solution parameters, click the **Global solution options** button. Be aware that the changes you make in the resulting dialog affect all power flow computations in Simulator.

### Output

The **Output** tab allows you to set a location for saving the results of the QV analysis to a file. To save the QV analysis to a text file, you must first check the box labeled **Save data in directory**. Once this box is checked, you can then either type in a directory location for saving the data, or click the **Browse** button and select a file to save as. You can also customize the file prefix and extension to be used when Simulator names the output files.

Another option on the **Output** tab is the check box labeled **Plot curves as they are computed**. When this option is checked, the QV curve displays will be updated and drawn during each step of the QV analysis process.

### Contingencies

The new QVCC can calculate QV curves for the specified buses for both base and contingency conditions. To analyze a set of contingencies, you must define the contingency set using the separate Contingency Analysis Form, which you can access from run mode by selecting **Options / Tools – Contingency Analysis** from the main menu.

To have QVCC analyze these contingencies for each bus, check the box labeled **Process each of the currently defined contingencies**.

## PVQV: Q-V Curve Control/Results

To perform the QV curve calculations, switch to the **Control/Results** tab and press the **Run** button. QVCC will respond by calculating QV curves for each of the specified scenarios (i.e. for each bus / contingency pair). When it finishes the QV curve calculation for a scenario, it will record the critical points for the curve in a case information display. These critical points include (V0, QV0), (Vmin, QVmin), (Vmax, QVmax), and (VQmin, Qmin).

Each scenario can be identified using the bus number and name fields together with the case name field, which simply specifies either "BASE CASE or the name of the contingency. As with all case information displays, the QV Results case information display has a local menu that offers a number of options. From the local menu, you can plot the QV curve for a scenario, record the (Q, V) pairs that comprise each curve to a text file or to an Excel spreadsheet, and clear the results.

The QVCC interface remains active while the curves are calculated. You can terminate the run at any time by clicking **Stop**.

## PVQV: Refine Model

Simulator PVQV has the ability to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during the PV and QV curve studies. This option is available from the main **Voltage Stability** menu.

The user can refine the case in the following ways:

### **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 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 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.

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 changed by right-clicking on it and selecting Toggle all to **YES/NO** or by double clicking on the field.

## Optimal Power Flow (OPF)

### Overview

#### **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 change 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 differentiates 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.

One important caveat is that the incremental impact of area losses **is not** considered during the enforcement of the area interchange constraint. This is similar to assuming that the bus penalty factors are all unity during an economic dispatch solution.

### 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.

One important caveat is that the incremental impact of super area losses **is not** considered during the enforcement of the super area interchange constraint. This is similar to assuming that the bus penalty factors are all unity during an economic dispatch solution.

### 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 equality 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 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 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 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 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 **can not** 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 practically always 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

Select **LP OPF**, **Primal LP** 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 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

- Solve the power flow

- Linearize the power system about the current power flow solution. Both constraints and controls are linearized.
- 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.
- Update the control variables and resolve the power flow.
- If the changes in the control variables are below a tolerance then the solution has been reached; otherwise go to step 2.
- Finish by resolving the power flow.

### OPF Primal LP Single Outer Loop

Select **LP OPF, Single Primal LP Outer Loop** to solve a single outer loop of the primal LP OPF algorithm. See OPF Primal LP for a description of the LP OPF solution algorithm. What this command does is just one loop through the algorithm (the jump back to step 2 is never executed). Thus this command allows you to manually perform an LP OPF solution. This can be helpful at times for figuring out what is going on during a particular OPF solution.

### OPF Future Enhancements

In this first version of Simulator OPF we have implemented the OPF functionality that will, we believe, be most useful to the largest number of users. 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.

In short 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. During the OPF solution, similar to what is done in the U.S. by the PJM interchange, the incremental impact of losses are not included. Thus, as was mentioned earlier, in the absence of binding inequality constraints all the bus marginal costs in an area will be equal.

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
- Optionally including the impact of area losses in the area/super area constraint enforcement.
- Including additional devices as controls, such as generator voltage setpoints, LTC transformers, switched shunts, dc line setpoints, load shedding.
- Including the impact of contingencies into the OPF solution.
- Allowing the optimization of different cost functions, such as maximization of social welfare.
- Additional functionality as suggested by customers.

We hope you have found the current version of Simulator OPF useful, and look forward to interacting with customers to help determine additional functionality for future versions.

## Options

### OPF Options

The OPF Options dialog allows you to customize the OPF solution. To display this dialog, select **LP OPF, Options**. The dialog consists of three general pages: General Options, Constraint Options, and Solution Results, two pages for users interested in viewing the details of the final LP basic variables and tableau: LP Basic Variables and LP Basis Matrix, and one page, Bus Marginal Price Details, for viewing marginal price information. Click on the corresponding tab at the top of the dialog to view the desired page.

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 LP OPF, Primal LP.

#### Single Outer Loop

Does a single outer loop of the Primal LP algorithm. Equivalent to selecting LP OPF, Single Primal LP Outer Loop.

#### Initialize LP OPF

#### 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 - 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.

#### **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.

#### **Limit Monitoring Settings...**

This button opens the Limit Monitoring Settings dialog, which allows you to change the enforcement percentages for monitored elements.

## **OPF Options - General Options**

The OPF Dialog, General Options page displays general options associated with the OPF solution. The display contains the following fields:

#### **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.

##### **Min. MW Constraint / MW Control Sensitivity**

Specifies the minimum absolute value of the sensitivity between a MW or MVA constraint and a MW control required before the MW control is allowed to be committed to remove the constraint violation. The purpose for this minimum is to prevent ineffective control actions. For example it usually makes little sense to make large changes in generator MW outputs when such changes have little impact of a particular constraint. Default = 0.01.

##### **Phase Shifter Sensitivity (MW / Degree)**

Specifies the minimum absolute value of the sensitivity between a MW or MVA constraint and a phase shifter control, expressed in MW / Degree before the phase shifter is allowed to be committed to remove the constraint violation. The purpose for this minimum is to prevent ineffective phase shifter control actions. Default = 1.000 MW / Degree.

##### **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.

#### **Controls**

##### **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.

#### **Objective Function**

Allows a choice of solving the LP using either a minimum cost or a minimum control change objective function.

#### **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 Modeling**

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.

### Case OPF Options File

This optional field is used to specify a default file name for storing OPF specific data. The OPF specific data is always stored with the pwb file. Additionally, in order to make it easy to transfer the OPF specific data between cases, this data may be stored in a pwo auxiliary file as well using the LP OPF, Store LP OPF Data command. The **Case OPF Options File** field specifies the default name for this file.

## 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 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 basic variables, showing the following fields for each:

#### **ID**

Basic variable identifier

#### **Cost**

Associated cost for the basic variable. This is the basic variable's entry in the LP cost vector.

#### **Constraint ID**

Constraint associated with the basic variable.

#### **Lambda**

Gives the marginal cost for enforcing the constraint.

## 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. Since the inverse of the basis matrix is also important, a button is provided at the top of the page to toggle between displaying the basis matrix and its inverse. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, right-click some where in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

## OPF Options - Solution Results

The OPF Dialog, Solution Results page displays general results from the last OPF solution. The display contains the following fields, none of which can be directly changed:

#### **General Results**

#### **Last 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 are 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.

**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.

**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 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.

**OPF Options - Bus Marginal Price Details**

The Bus Marginal Price Details page displays a grid containing the 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.

**OPF Records**

**OPF Bus Records**

Displays OPF specific information about each bus record with a valid area/zone filter. To show this display select **LP OPF, OPF Bus Records**. 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 99,999, and its alphanumeric identifier, eight characters maximum.

**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 filter. To show this display select **LP OPF, OPF Generator Records**. 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. 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

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

Single character ID used to distinguish multiple generators at the same bus.

### Area Name

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 (see following).

### 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 new real power output of the generator at the conclusion of the OPF solution.

### 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.

### 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.

### **Delta MW**

Change in the generator's real power output 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 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 only 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.

## **OPF Line/Transformer Records**

Displays OPF specific information about the line and transformer records in the case. To show select **LP OPF, OPF Line/Transformer Records**. 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 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. 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**

Number and Name of the From bus.

### **To Number, To Name**

Number and Name of the To bus.

### **Circuit**

Circuit identifier for the branch.

### **Enforce MVA**

Specifies whether or not the branch's 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.

**Max Percent**

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**

The marginal cost of changing the MVA flow on the branch.

**MVA Unenforceable**

This field will be set to YES if the branch limit is unenforceable in the OPF solution.

**Constraint**

This field will be set to YES if the branch limit is a constraint in the OPF solution.

## OPF Area Records

Displays OPF specific information about each area record in the case. To show this display select **LP OPF, OPF Area Records**. 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. 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**

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 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 Dialog, and the individual interfaces must be enabled for enforcement on the OPF Interface MW Constraints display.

**Include Marg. Losses**

Specifies whether or not marginal losses should be included for the area during the OPF solution.

**Avg. Lambda**

For a 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.

**Lambda St.Dev., Min. Lambda, Max. Lambda**

For a 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.

## **OPF Super Area Records**

Displays OPF specific information about each super area record in the case. To show this display select **LP OPF, OPF Super Area Records**. 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. The columns can also be sorted by right-clicking on the heading of the field. By default the display contains the following fields:

**Name**

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.

**Avg. Lambda**

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.

**Lambda 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.

**Min Lambda, Max Lambda**

Minimum and maximum of all the bus MW marginal costs for the buses in the super area.

**OPF Interface Records**

Displays OPF specific information about the interface records in the case. To show select **LP OPF, OPF Interface Records**. 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 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. The columns can also be sorted by right-clicking on the heading of the field. By default the Interface Records page contains the following fields:

**Name**

Name of the interface.

**Enforce MW**

Specifies whether or not the interface MW limit is enforced in the OPF solution.

**MW Flow**

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.

**MW Marg. Cost \$ / MWh**

The marginal cost of enforcing the limit on the interface.

**MW Unenforceable**

This field will be set to YES if the interface limit is unenforceable in the OPF solution.

**Constraint**

This field will be set to YES if the interface limit is a constraint in the OPF solution.

## Controls

### 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 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

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.

#### Phase shifting Transformer tap position (*not quite done*)

Phase shifting transformers are used primarily to control the flow of real power in the network.

*Additional details on the use of phase shifters will be provided shortly.*

### OPF Phase Shifter Records

Displays OPF specific information about each phase shifter record in the case. To show this display, select **LP OPF, OPF Phase Shifter Records** from the main menu. 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.

#### 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 Restore Previous Control Settings

Select **LP OPF, Restore Previous Control Settings** to restore the control settings to what they were before the last OPF solution. This menu item is only available following an OPF solution. After doing this command you will also need to resolve the power flow to restore the previous case.

## Examples

### OPF Example - Introduction

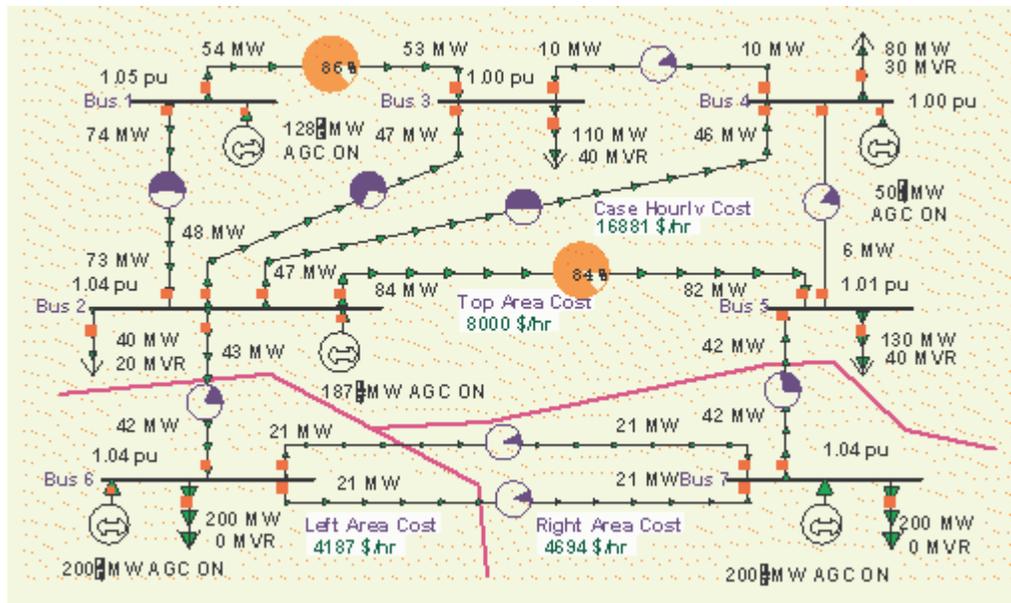
As a simple example of using the OPF, consider the seven bus, three area system contained in the file B7FLATOPF (included with the PowerWorld Simulator). For this case all three areas are initially on Economic Dispatch (ED) AGC control and hence by default would not be included in the OPF solution. Also, 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**. The case should look like the first figure below.

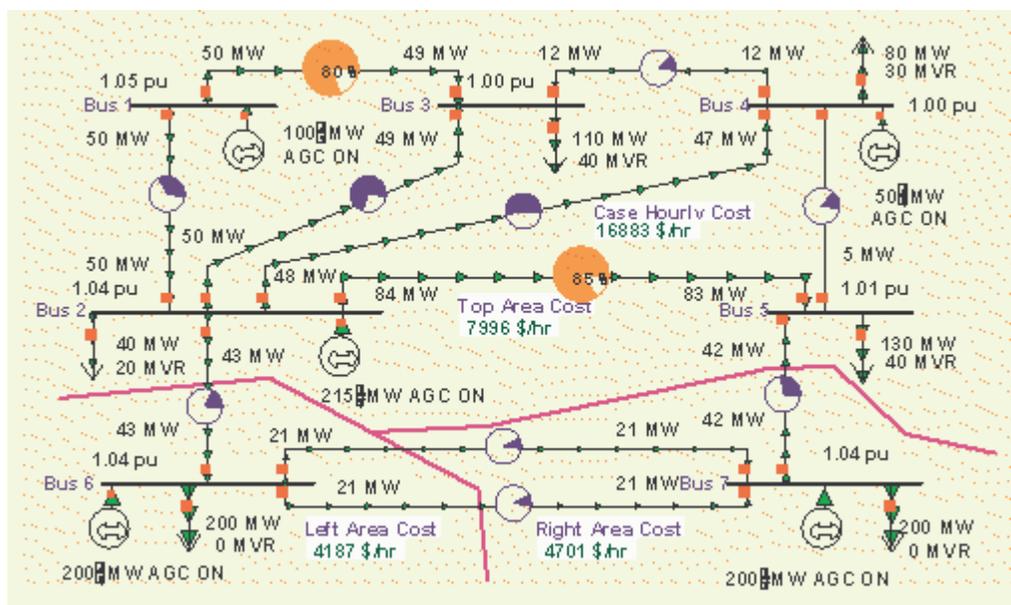
Now we'll modify the case to set the three areas for OPF control. To do this, select **LP OPF, OPF Area Records** to display the OPF Area Records display. Toggle the AGC status for each of the three areas to change it to "OPF". Now select **LP OPF, Primal LP** to solve the case using the LP OPF. The results should look similar to the second figure below. The one-line shows the hourly cost for each area and the total case hourly cost, equal to \$ 16,883 / hr.

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 General Options page of the OPF Options display.

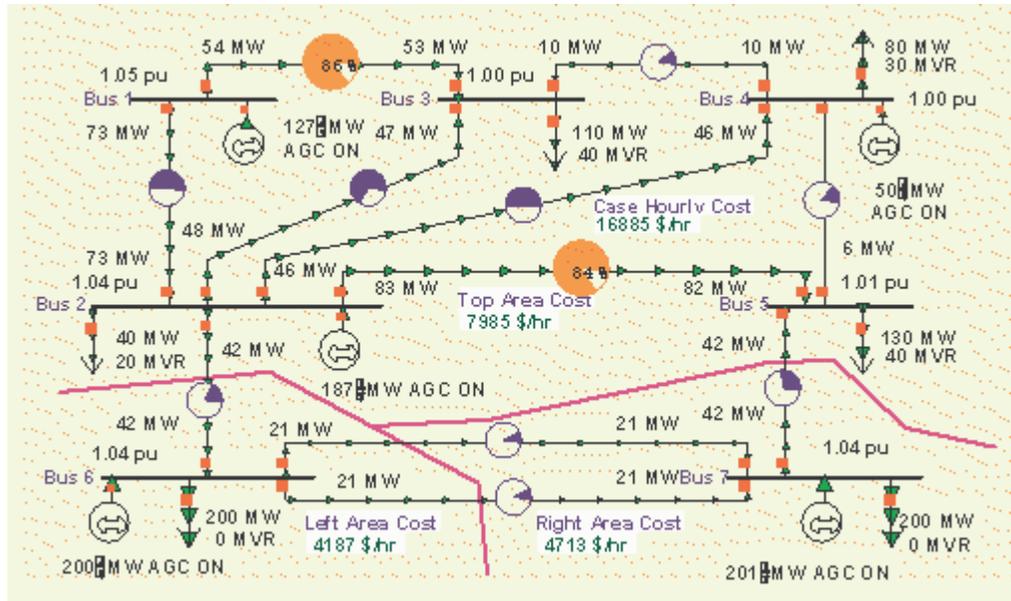
Change the **Total Points Per Cost Curve** field to 100 and resolve. The results are shown in the third figure below, which now are almost identical to the economic dispatch results. The disadvantage though in using a large number of cost segments is that it degrades the performance of the LP OPF slightly on larger cases.



B7OPF Case Solved using Economic Dispatch



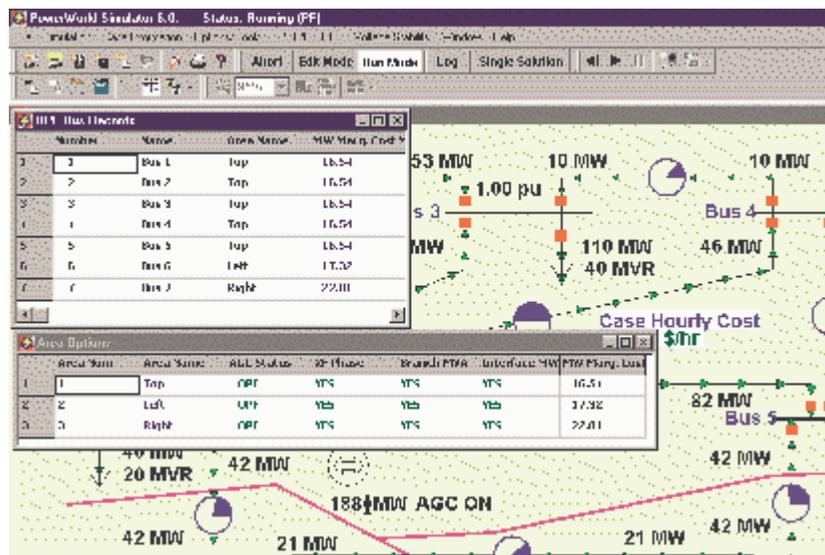
B7OPF Case Solved using LP OPF



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 LP OPF, Bus Records to view the bus marginal costs and LP OPF, OPF Area Records to view the area marginal costs. The results should be as shown below.



Seven Bus Case Bus and 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.

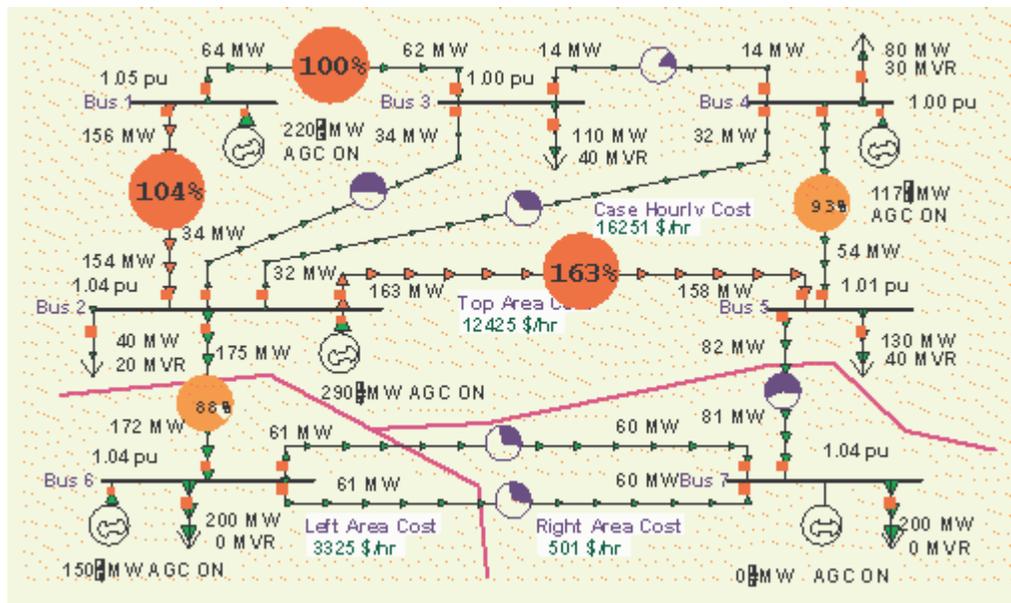
## OPF Example - Super Areas

To jointly dispatch the three areas we'll first combine them into a single super area. To setup the super area first select **LP OPF, OPF Super Area Records** to display the OPF Super Area Records display (alternatively you could also use the **Case Information, Super Area** 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. In the **Name** field enter a name for the new super area, "ThreeAreas." Then select the **Add New** button to create a new super area. 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.

Before resolving the OPF lets temporarily disable enforcement of line MVA constraints. You can do this from the Constraint Options page of the OPF Options 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 **File, Save Case As** to save the case (pwb and pwr files) with a different name, say B7OPFSA (SA for super area).

Select **LP OPF, Primal LP** to resolve the OPF. The results should be as shown below.



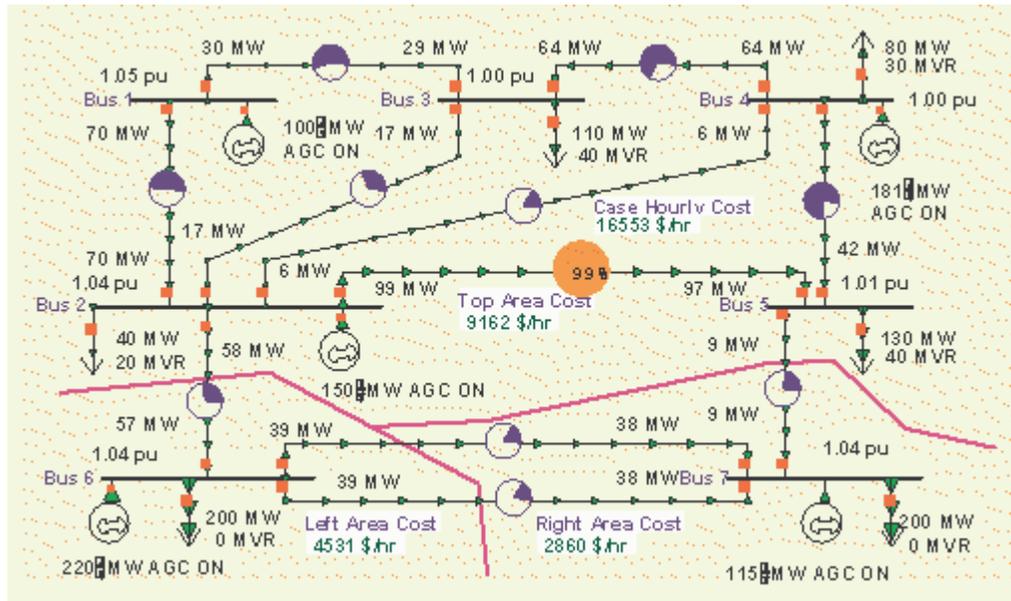
OPF Solution with Super Area WITHOUT Enforcing Line MVA Constraints

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 \$ 16,885 / hr to \$ 16,251 / 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 LP OPF, OPF Options 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

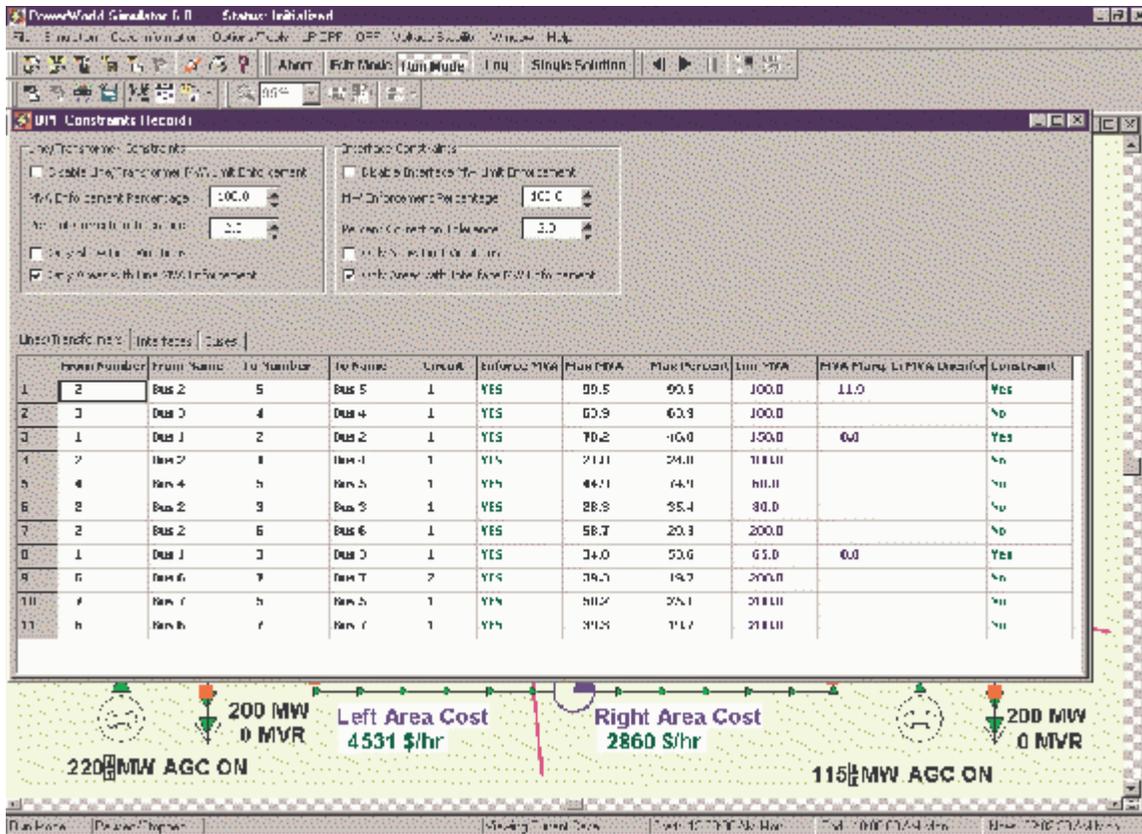
With line constraint enforcement active the OPF optimally redispatches the generation taking into account the line MVA limits. However enforcing these line constraints comes at a cost. Notice that the total case hourly cost has increased from \$ 16,251 / hr to \$ 16,553 / hr, which is still substantially less than the \$ 16,885 / 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.

OPF Bus Records					
	Number	Name	Area Name	MW Marg. Cost	MVR
1	1	Bus 1	Top	16.19	
2	2	Bus 2	Top	15.90	
3	3	Bus 3	Top	17.08	
4	4	Bus 4	Top	17.44	
5	5	Bus 5	Top	22.28	
6	6	Bus 6	Left	17.45	
7	7	Bus 7	Right	20.60	

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 LPOPF, OPF Line/Transformer MVA Records display. The MVA Marg. Cost tells the marginal cost of enforcing the constraint, expressed in units of \$ / MVA / hr. Right-click on the **MVA Marg Cost** field header to sort the display using this field.



## OPF Example Page 5

More to come!

We hope that this brief OPF example has been helpful. Please let us know if you have any questions or suggestions.

## Storing OPF Data

### Load LP OPF Data

Select **LP OPF, Load LP OPF Data** to load LP OPF specific fields into the case from the specified \*.pwo text file. Note, the LP OPF specific fields are saved with the case in the pwb file. The pwo files should therefore only be used for transferring the opf specific data between cases or for the user to manually change the data using a text editor. Existing OPF specified data can be stored in a \*.pwo text file using the LP OPF, Store LP OPF Data menu command.

### Store LP OPF Data

Select **LP OPF, Store LP OPF Data** to store LP OPF specific fields from the case into the specified \*.pwo text file. Note, the LP OPF specific fields are saved with the case in the pwb file. The pwo files should therefore only be used for transferring the opf specific data between cases or for the user to manually change the data using a text editor. Data can be loaded into a case from the \*.pwo text file using the LP OPF, Load LP OPF Data menu command.

# Available Transfer Capability (ATC) Analysis

## Available Transfer Capability (ATC) Analysis

Available Transfer Capability (ATC) analysis determines the maximum MW transfer possible between two parts of a power system without violating any limits. Most often, this transfer is between two areas in the system. For information on how to use the Simulator ATC tool, see Available Transfer Capability Dialog.

Simulator's ATC analysis makes use of several tools that are available elsewhere in Simulator. These include

- Power Transfer Distribution Factors (PTDFs): determines the linear impact of a transfer (or changes in power injection) on the elements of the power system.
- Line Outage Distribution Factors (LODFs): determines 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: controls 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 these same 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 output of the ATC analysis tool.

Simulator provides three methods of determining the ATC for a transfer direction, which are described below.

### Single Linear Step (SL)

The Single Linear Step approach is the most common ATC method and duplicates the ATC analysis done by many system reliability organizations throughout North America. 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 and present loading of 5 and a PTDF of 10%. The estimated maximum transfer without causing an overload on line is

$$\text{Transfer Limitation} = (\text{Limit} - \text{Present Loading}) / \text{PTDF} = (10 - 5) / 0.1 = 50 \text{ MW}$$

When including contingency analysis, then 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 Limitation.

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.

### 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, but still performs its analysis in a reasonable period of time. The (IL) method operates as follows

- Stepsize = ATC using Single Linear Step (Note: save this initial step size for use by Full AC)
- If  $[\text{abs}(\text{stepsize}) < \text{Tolerance}]$  then stop
- Ramp transfer out an additional amount of Stepsize and resolve Power Flow
- At new operating point, Stepsize = ATC using Single Linear Step
- Go to step 2

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.

#### Iterated Linear Step (IL) then Full AC

If you want to absolutely test the validity of the ATC number, then you can use this method. Be aware however, that this calculation method can be extremely slow. The (IL) then Full AC method operates as follows.

- Run Iterated Linear Step and ramp transfer out ATC Value found
- StepSize = 10% of the initial Linear Step Size saved during the (IL) method, or 50 MW whichever is larger.
- Run Full Contingency Analysis on the ramped transfer state
- If there are violations then change the sign of Stepsize
- if  $[\text{abs}(\text{stepsize}) < \text{Tolerance}]$  then Stop
- Ramp transfer out an additional amount of Stepsize and resolve Power Flow
- At new operating point, Run Full Contingency Analysis
- if  $[(\text{Stepsize} > 0) \text{ and } (\text{There are Violation})]$  OR  $[(\text{Stepsize} < 0) \text{ and } (\text{There are NO Violations})]$  THEN  
StepSize := -StepSize/2 (switch direction)
- Go to step 5

## Available Transfer Capability Dialog

The Available Transfer Capability Dialog provides you 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. For information on performing analysis on multiple scenarios please see the Multiple Scenario Available Transfer Capability Dialog.

When performing analysis on only one scenario the ATC Dialog has three tabs: **Options**, **Analysis**, and **Result**. There are many settings on these tabs that will be described here. If you have setup your analysis in a manner you would like to repeat in the future, click on Save Settings to save your settings in a file. Then when you come back into the dialog at a later time, click Load Settings and retrieve the file containing your settings.

The three tabs are described now.

#### Options Tab: Common Options

##### 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, participation groups, buses, or to a slack bus. For other ATC Solution Methods, ATC can be calculated for transfers between areas and super areas 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. Their contents are filled when you select the seller and buyer types. You can also use Simulator's advanced search engine to find the buyer and seller by clicking on the **Find Seller...** and **Find Buyer...** buttons.

#### **Reverse Buyer/Seller**

Click this button to reverse of 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 is a modification to the lossless dc approximation that takes into account phase shifter operation. It is especially useful when the ATC tool keeps reporting overloads on branches that obviously will not overload because of the operation of a phase shifting transformer.

#### **Include Contingencies**

Check this box to include contingencies previously inserted or loaded in the Contingency Analysis Tool in the 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.

#### **Limit Monitoring Settings**

Click this button to open the Limit Monitoring Settings Dialog. Note that minimizing the number of monitored power system elements greatly improves solution speed as well as computer memory requirements for doing ATC Analysis.

#### **Transfer Result Reporting Options**

##### **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 pieces of information

- Transfer Limit in MW
- Transmission branch (or interface) that causes the limit
- Contingency that is applied to cause the limit (if it's 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**

Also, when analyzing a long list of contingencies, the worst transfer limitations may all be overload 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.

**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 100 MW transfer, there would be only a 0.5 MW increase in flow on the Limiting Element.

As described in Available Transfer Capability (ATC), the transfer limitation functions involve dividing by the PTDF or OTDF values for each branch or interface. 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 anyway.

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.

**Options Tab: Advanced Options**

**ATC Solution Method**

Simulator provides three methods of determining the ATC for a transfer direction.

- Single Linear Step (SL)
- Iterated Linear Step (IL)
- (IL) then Full AC Solution

These methods were described in detail in the explanation of Available Transfer Capability (ATC).

**Transfer Calculation Options**

**Transfer Tolerance**

This is the tolerance used using the Iterated Linear or (IL) then Full AC Solution Methods.

**When iterating, Ignore Limiters below**

This value indicates a minimum transfer value for reporting the ATC analysis results. Any ATC results below this value will not be displayed. In general, you do not want to use this option, as you are interested in all transfer limitations during various contingencies. However, sometimes you may wish to ignore ATC results below 0, as a negative ATC result signifies a base case overload or limitation under the given contingency condition, prior to any additional transfer in the defined seller to buyer direction.

**Enforce OPF Constraints**

This only affects the Iterated Linear or (IL) then Full AC Solution Methods. When include OPF Constraints is checked, as the solution process ramps the transfer up, it will also solve the OPF after each transfer increase.

#### **Model reactive power for linear methods by...**

The linearized methods used in ATC 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 MVA flows during the linear calculations. The choices are **Ignoring reactive power**, **Assuming constant voltage magnitude**, or **Assuming reactive power does not change**.

#### **Define Extra Monitors**

Simulators ATC tool determines the maximum amount of MW transfer between the buyer and seller. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, then you can utilize the Extra Monitors. Click on this button to open the ATC Extra Monitors Dialog which can be used to define the Extra Monitors.

#### **Analyze Multiple Scenarios**

Check Analysis Multiple Scenarios to perform ATC Analysis on several scenarios. See ATC Dialog For Multiple Scenarios for more information on Multiple Scenario Analysis.

#### **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.

#### **Analysis Tab**

##### **Start Analysis**

Click this button to begin the analysis.

##### **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 log.

##### **Restore Initial State**

This will restore the system state to the state when the dialog was first opened.

##### **Increase Transfer**

Click this button to increase the transfer level manually.

#### **Result Tab**

When performing ATC Analysis on a single system state, the Result Tab is visible. The Result Tab contains a list of the Transfer Limiters that were found by the analysis. See Transfer Limiters Display for more information.

## ATC Extra Monitors Dialog

Simulator's ATC tool determines the maximum amount of MW transfer between the buyer and seller. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, then you can utilize ATC Extra Monitors. The ATC Extra Monitors Display contains a list display that lists all the ATC Extra Monitors defined. This list display is a Case Information Display, and therefore has the same functionality as other common

displays. To insert an Extra Monitor, right-click on the list display and choose insert. Then choose whether you want to monitor an interface MW flow or a Branch MW flow. Finally choose the interface or branch you are interested in. To delete an extra monitor, right-click on the list display and choose delete.

The default fields shown on this display are

### **ATC ExMon Desc**

This is a description of the value that is being monitored. Presently, this is always MW flow.

### **ATC ExMon Obj**

The power system element that is being monitored. This will either be 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, then only Transfer Limitations which meet the following condition are included in the results.

$$\frac{OTDF_{ELEMENT}}{PTDF_{EXTRAMON}} \times \frac{ExtraMon Rating}{Element Rating} \geq Relative Monitor$$

This provides a measure of how much an interface or branch is affected by the transfer relative to its MW limit.

## Multiple Scenario Available Transfer Capability Dialog

To perform Available Transfer Capability Analysis for several system scenarios, check Analyze Multiple Scenarios on the Available Transfer Capability Dialog. When Analyze Multiple Scenarios is not checked the available tabs are Options, Analysis and Result, which are described in the topic Available Transfer Capability Dialog. When Analyze Multiple Scenarios is checked then the following changes occur:

- Scenarios Tab appears
- Results Tab appears
- Result Tab is removed (replaced by Results)

The other tabs remain the same, so help regarding them can be found with the Available Transfer Capability Dialog help.

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

Once you have specified these scenarios, then Simulator is able to perform ATC Analysis on every combination of these 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  $10 \times 8 \times 3 = 240$  different scenarios to calculate ATC for. Be warned that the more scenarios you analyze the longer the computer computation will take.

The new tabs Scenarios and Results allow you to define these scenarios and view their results.

#### **Scenarios Tab**

The Scenarios Tab has three “subtabs:

- Line Ratings/Zone Loads
- Generator Outputs
- Interface Ratings

Each tab contains a list of the power system elements that will be modified during different scenarios. This list is a familiar Case Information Display providing the same functionality as other displays by right-clicking to bring up the local menu. To insert a new power system element into the list, right click on the list (below the headings) and choose Insert.

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.

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. On the Line Ratings/Zone Loads tab, notice that a tab appears toward the bottom of the dialog that contains two options: Line Ratings and Zone Loads. This tab toggle between the list displays for lines and zones.

#### **Results Tab**

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 “workbook display. 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 IO, IO, ... for the Interface Scenarios.

To change the order of these axis change Axis Order.

To view the Transfer Limiters found under each scenario, click the Show Transfer Limiters button. 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.

If you right click on the workbook, you will bring up a local menu containing several options. The first several options are only enabled if you are right click on a cell representing a scenario. These options will perform an operation with respect to the Scenario:

#### **Take me to Scenario ...**

Will make modifications to the system state to take you to the Scenario.

#### **Determine Transfer Limit For Scenario ...**

Will calculate the ATC for the Scenario, and then take you back to the Initial State.

Take me to the Transfer Limit For Scenario ...

Will perform ATC for the Scenario, and then ramp the transfer to this limit.

Other options on this local menu are not related to the Scenario you have clicked on.

**Increase Transfer for Present System State**

Will increment the transfer level for the present system state by a user-defined amount.

**Return to Initial State**

Will return the system state to the Initial State.

## Transfer Limiters Display

The Transfer Limiters Display appears on the Result Tab of the Available Transfer Capability Dialog or as a separate window when performing Multiple Scenario Available Transfer Capability Analysis. This display shows the results of ATC Analysis.

The display contains three different tabs: All Limiters, Branch Limiters, and Interface Limiters. The All Limiters tab shows a list of all the transfer limitations found. This includes limitations on branches, interfaces, areas, zones, etc... 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. 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.

Lastly, a Contingency Definition Display is also available at the bottom of the Result Tab. This list will display the defined contingency that caused the selected limitation from one of the three pages of limiters.

The fields that are included on this display are

**Transfer Limit**

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 Limitation values are those found during this step.

**Iterated Linear Step (IL)**

The Linear ATC method is iterated during this method. The Transfer Limiters shown are those found during the final step performed. The actual Transfer Limitation values are the values found at the last step plus the accumulated amount the transfer has been ramped. The transfer limitation that was used when iterating is highlighted in yellow.

**Iterated Linear Step (IL) then Full AC**

The Transfer Limiters shown are those found during the last contingency analysis step that found violations. The Transfer Limitation values equal the accumulated amount the transfer was ramped. Note that when performing the Full AC, negative ATC values will not be probed, so the Transfer Limit will appear as 0 if there are violations in the Initial State.

**Limiting Element**

Shows a text description of the limiting element

**Limited CTG**

Shows the name of the limiting contingency

**% OTDF**

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 Linear, this is the PTDF or OTDF at the last Linear Iteration. When using Full AC Solution Method, this is not calculated and will appear as 0 for all Limiters.

**Initial Value**

This is the flow on the Limiting Element in the Initial State (before any transfer or contingencies occur).

**Limit Used**

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.

**Pre-Transfer Value Estimate**

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 Linear, this is the estimate at the last Linear Iteration. When using (IL) Full this is not calculated and will appear as 0 for all Limiters.



## 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 Script and Data Files, including how they are created and loaded in Simulator
- Auxiliary Script section format
- Auxiliary Data section format

### 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 auxiliary files in Simulator. 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)**

```
{
data_list_1
.
data_list_n
}
```

```
}
```

**DATA DataName2(object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
```

```
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 Section

### Script Section

The **SCRIPT** section begins with a left curly brace and ends with a right curly brace.

**SCRIPT**

```
{
```

```
script_statement_1
```

```
script_statement_2
```

```
.
```

```
.
```

```
.
```

```
script_statement_n
```

```
}
```

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 an 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.

Those familiar with using Simulator will know that there is a RUN and EDIT mode in Simulator. Some features in Simulator are only available in one mode or the other. This functionality will be preserved in the script language, but additionally a feature called a “submode will be used.

Submodes limit what script commands can be called. Only those commands available to the submode can be executed. To switch submodes, use the EnterMode (mode, submode) script command.

You will always be in one of the submodes when executing a script. By default, when a script is initially started, you will be placed in the **RUN, POWERFLOW** submode.

## Script General Actions

### Script General Actions

The following actions are available to you regardless of the Mode or SubMode:

LoadAux ("filename", CreateIfNotFound);

LoadScript ("filename", ScriptName);

LoadData ("filename", DataName, CreateIfNotFound);

SelectAll (objecttype, filter);

UnSelectAll (objecttype, filter);

Delete (objecttype, filter);

SaveData ("filename", filetype, objecttype, [fieldlist], [subdatalist], filter);

SetData (objecttype, [fieldlist], [valuelist], filter);

CreateData (objecttype, [fieldlist], [valuelist]);

ChangeData (objecttype, [fieldlist], [valuelist], filter); (NOT AVAILABLE YET)

```

NewCase;

OpenCase ("filename");// assumes to open as PWB

OpenCase ("filename",    openfiletype );

                        PWB

                        GE

                        PTI

                        IEEECF

SaveCase ("filename"); // assumes save as PWB

SaveCase ("filename",    savefiletype);

                        PWB, PWB5, PWB6, PWB7,

                        PTI23, PTI24, PTI25, PTI26, PTI27,

                        GE

                        IEEECF

EnterMode (mode, submode);

                        EDIT  CASE

                        RUN  POWERFLOW

                        CONTINGENCY

                        ATC

                        FAULT

                        PV

                        QV

```

**LoadAux("filename, CreatelfNotFound);**

Use this action to load another auxiliary file from within a script.

**"filename :** The filename of the auxiliary file being loaded.  
**CreatelfNotFound :** Set to YES or NO. YES means that when reading DATA sections, objects which can not be found will be created while reading in DATA sections.

**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, CreatelfNotFound);**

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.  
**CreteIfNotFound :** Set to YES or NO. YES means that when reading DATA sections, objects which can not be found will be created while reading in DATA sections. 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, "filename"); :** "filename means select those that meet the filter  
**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 properties to false.

**Delete(objecttype, filter);**

Use this delete objects of a particular type. A filter may optionally be specified to only delete object which 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, "filename"); :** "filename means delete those that meet the filter  
**Delete(objecttype, AREAZONE); :** AREAZONE means delete those that meet the area/zone filters  
**Delete(objecttype, SELECTED); :** SELECTED means delete those that are selected

**SaveData("filename, filetype, objecttype, [fieldlist], [subdatalist], filter);**

Use this action to save data in a custom defined format. A filter may optionally be specified to save only object which meet a filter.

**"filename :** The file to save the data to  
**filetype :** AUX (or AUXCSV) – save as a space-delimited (or comma-delimited) auxiliary data file.  
**objecttype :** the objecttype being saved  
**[fieldlist] :** A list of fields that you want to save  
**[subdatalist] :** A list of the subdata objecttypes to save with the  
**filter :** There are four options for the filter:  
**SaveData(...); :** No filter specified means to save all objects of this type.  
**SaveData(..., "filename); :** "filename 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 that are selected

**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 filter

**SetData(..., AREAZONE);** : AREAZONE – set data for all objects that meet the area/zone filters

**SetData(..., SELECTED);** : SELECTED – set data for all objects that are selected

**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

**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, IEEECF.

**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  
PTI23, PTI24, PTI25, PTI26, PTI27  
GE  
IEEECF

**EnterMode(mode or submode);**

This action will save part of the power system to a "filename. It will save only those buses whose property pwEquiv must is set true.

**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.

## Script Edit Mode Actions

### 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]);

LOAD

MW

[P,Q]

```

GEN          FACTOR          [P] (means constant power factor
INJECTIONGROUP

BUSSHUNT    MW              [P, +Q, -Q]
            FACTOR

Move        ([elementA],    [destination parameter]);
            [GEN numA idA]    [numB idB]
            [LOAD numA idA]   [numB idB]                // NOT
AVAILABLE YET

            [BRANCH numA1 numA2 cktA] [numB1 numB2 cktB]    // NOT
AVAILABLE YET

Combine([elementA],        [elementB]);
            [GEN numA idA]    [GEN numB idB]
            [LOAD numA idA]   [LOAD numB idB]                // NOT
AVAILABLE YET

            [BRANCH numA1 numA2 cktA] [BRANCH numB1 numB2 cktB] // NOT
AVAILABLE YET

```

**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 `pwEquiv` 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 `pwEquiv` 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 `pwEquiv` 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`  
`PTI23, PTI24, PTI25, PTI26, PTI27`  
`GE`  
`IEEECF`

**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 `pwEquiv` 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]);**

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

**Move([elementA], [destination parameters]);**

NOTE: THIS ACTION IS ONLY AVAILABLE FOR GENERATORS IN VERSION 8.0

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]  
 [LOAD busnum id]  
 [BRANCH busnum1 busnum2 ckt]

**[destination parameters] :** These parameters have different meanings depending on object type of the element.

GEN - [busnum id]  
 LOAD – [busnum id]  
 BRANCH – [busnum1 busnum2 id]

**Combine([elementA], [elementB]);**

NOTE: THIS ACTION IS ONLY AVAILABLE FOR GENERATORS IN VERSION 8.0.

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

## Script Run Mode Actions

### Script Run Mode Actions

The following script commands are available during any of the submodes of Run Mode:

CalculatePTDFs ([transactor seller], [transactor buyer],  
 LinearMethod);

AC [AREA num] [AREA num]

DC [ZONE num] [ZONE num]

```

DCPS [SUPERAREA "name"] [SUPERAREA name]

name] [INJECTIONGROUP "name"] [INJECTIONGROUP

[BUS num] [BUS num]

[SLACK] [SLACK]

CalculateLODFs ([BRANCH nearbusnum farbusnum ckt], LinearMethod);

CalculateTLRs ([flowelement], direction, [transactor],
LinearMethod);

for PTDFs [INTERFACE "name"] BUYER same as above

[BRANCH nearbusnum farbusnum ckt] SELLER

CalculateVoltSense ([BUS num]);

CalculateFlowSense ([flowelement], FlowType);

[INTERFACE "name"] MW

[BRANCH num1 num2 ckt] MVAR

MVA

```

**CalculatePTDFs([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]**

**[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

DCPC – for lossless DC that takes into account phase shifter operation

**CalculateLODFs([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.

**LinearMethod** : the linear method to be used for the LODF calculation. The options are  
 DC – for lossless DC  
 DCPC – for lossless DC that takes into account phase shifter operation

Note: AC is NOT an option for the LODF calculation

**CalculateTLRs([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]  
 [BRANCH nearbusnum farbusnum ckt]

**direction** : the type of the transactor. Either BUYER or SELLER.

**[transactor buyer]** : the transactor of power. There are six possible settings  
 [AREA num]  
 [ZONE num]  
 [SUPERAREA "name]  
 [INJECTIONGROUP "name]  
 [BUS num]  
 [SLACK]

**LinearMethod** : the linear method to be used for the calculation. The options are  
 AC – for calculation including losses  
 DC – for lossless DC  
 DCPC – for 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 an 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]  
 [BRANCH num1 num2 ckt]

**FlowType** : The type of flow to calculate this for. Either MW, MVAR, or MVA.

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;
```

```
SolvePrimalLP;
```

```
DiffFlowSetAsBase;
```

```
DiffFlowClearBase;
```

```
DiffFlowMode      (diffmode);
```

```
    PRESENT
```

```
    BASE
```

```
    DIFFERENCE
```

```
OPFWriteResultsAndOptions ("filename");
```

**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;**

Call this action to solve the power flow.

**SolvePrimalLP**

Call this action to perform a primal LP OPF solution.

**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
```

**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.

## Script Contingency Submode Actions

The following actions are specific to the Run Mode Submode Contingency:

```
CTGSolveAll;
```

```

CTGSolve("ContingencyName");

CTGSetAsReference;

CTGRestoreReference;

CTGProduceReport("filename");

CTGWriteResultsAndOptions("filename");

```

**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.

**Script ATC Submode Actions**

The following actions are specific to the Run Mode Submode ATC:

```

ATCDetermine      ([transactor seller],      [transactor buyer]      );
                  [AREA num]                [AREA num]
                  [ZONE num]                [ZONE num]
                  [SUPERAREA "name"]        [SUPERAREA name]
                  [INJECTIONGROUP "name"]    INJECTIONGROUP
name]
                  [BUS num]                 [BUS num]
                  [SLACK]                   SLACK]

```

```

ATCRestoreInitialState;

```

```

ATCIncreaseTransferBy      (amount);

```

```

ATCTakeMeToScenario      (RL, G, I);

```

```

ATCDetermineFor      (RL, G, I);

```

ATCWriteResultsAndOptions ("filename");

**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]

[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.

**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.

**ATCDetermineFor(RL, G, I);**

Call this action to determine the ATC for Scenario RL, G, I.

**ATCWriteResultsAndOptions("filename);**

Writes out all information related to ATC analysis as an auxiliary file. This includes Contingency Definitions, Limit Monitoring Settings, Solution Options, ATC Options, ATC results, as well as any Model Criteria that are used by the Contingency Definitions.

## Script Fault Submode Actions

The following actions are specific to the Run Mode Submode Fault:

Fault ([BUS num], faulttype, R,  
X);

Fault ([BRANCH nearbusnum farbusnum ckt], faultlocation, faulttype,  
R, X);

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.  
**[BRANCH nearbusnum farbusnum ckt]** : this specifies the branch on which the fault occurs.  
**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 specified 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:

PVCalc([list-of-contingencies]); // calc pv curves for a set of ctgs. If none, base case

SELECTED

NOTSKIPPED

PVSaveResults("filename");

PVSavePlot ([BUS num], [CONTINGENCY ContingencyName], "filename",  
 imagefiletype);

WMF

JPEG

BMP

**PVCalc([contingencies]);**

Call the function to calculate a PV curve. You may optionally specify a list of contingencies to use as well. If no contingencies are specified the base case will be assumed.

**[contingencies]** : this describes which contingencies you would like to include in the PV curve calculation. There are three options for specifying the list (1) include a comma-separated list of contingency names, (2) choose all contingencies marked SELECTED, or (3) choose all contingencies not marked SKIP.

["contingency name 1, "contingency name 2, ... ]

SELECTED

NOTSKIPPED

**PVSaveResults("filename");**

Call this action to save all the PV results in the file "filename.

**PVSavePlot([BUS num], "Contingency Name", "filename, imagefiletype);**

Call the function to calculate a PV curve. You may optionally specify a list of contingencies to use as well. If no contingencies are specified the base case will be assumed.

**[BUS num]** : the bus

**"Contingency Name** : The contingency

**“filename :** The file to save the image in  
**imagefiletype :** The kind of image to create and save. There are three options  
 WMF  
 JPEG  
 BMP

## Script QV Submode Actions

The following actions are specific to the Run Mode Submode QV:

```

QVCalc ([list-of-buses], [list-of-contingencies]); // calculate qv curves for a
// list of buses and list-of-contingencies
// if not contingencies, then use base case

```

```

SELECTED SELECTED
NOTSKIPPED

```

```
QVSaveResults("filename");
```

```

QVSavePlot ([BUS num], [CONTINGENCY ContingencyName], "filename",
imagefiletype);

```

```
WMF
```

```
JPEG
```

```
BMP
```

**QVCalc([buses], [contingencies]);**

Call the function to calculate a QV curve for a list of buses. You may optionally specify a list of contingencies to use as well. If no contingencies are specified the base case will be assumed.

**[buses] :** this describes the list of buses to calculated QV curves for.  
**[contingencies] :** this describes which contingencies you would like to include in the QV curve calculation. There are three options for specifying the list (1) include a comma-separated list of contingency names, (2) choose all contingencies marked SELECTED, or (3) choose all contingencies not marked SKIP.

```
["contingency name 1, "contingency name 2, ... ]
```

```
SELECTED
```

```
NOTSKIPPED
```

**QVSaveResults("filename");**

Call this action to save all the QV results in the file “filename.

**QVSavePlot([BUS num], “Contingency Name, “filename, imagefiletype);**

Call the function to calculate a PV curve. You may optionally specify a list of contingencies to use as well. If no contingencies are specified the base case will be assumed.

**[BUS num] :** the bus

**“Contingency Name :** The contingency

**“filename :** The file to save the image in

**imagefiletype :** The kind of image to create and save. There are three options  
 WMF  
 JPEG

## Data Section

### 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 (object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
data_list_1
.
.
.
data_list_n
}
```

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 “( )” 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)	Data fields are space delimited
CSV	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 BUS, then the data describes BUS objects. Simulator currently recognizes the following object types:

Area	DCTransmissionLine	PartPoint
ATC_Options	Direction	PVCurve_Options
ATCExtraMonitor	Equip_Options	QVCurve_Options

ATCGeneratorChange	Filter	ReactiveCapability
ATCInterfaceChange	Gen	Scale_Options
ATCLineChange	InjectionGroup	Schedule
ATCScenario	Interface	Shunt
ATCZoneChange	InterfaceElement	Sim_Environment_Options
BidCurve	Limit_Monitoring_Options	Sim_Misc_Options
Branch	LimitSet	Sim_Simulation_Options
Bus	LimitViol	Sim_Solution_Options
ColumnInfo	Load	StudyMWTransactions
Condition	LoadScale	Substation
Contingency	ModelCondition	SuperArea
Ctg_Options	ModelFilter	TransferLimiter
CTGElement	MSLine	Transformer
CustomExpression	MWTransaction	XFCorrection
DataGrid	OPF_Options	Zone
	OwnerObject	

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 main menu and choosing **Help > 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 800 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 main menu and choosing **Help > 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, [pwBusNomKV, pwBus, // comment here

pwABCPhaseAngle:1, pwABCPhaseAngle:2, pwABCPhaseV, pwABCPhaseV:1,

// comments allowed here to

// note that blank rows are ignored

pwAreaNum, pwBusAngle, pwBusB, pwBusCat, pwBusEquiv, pwBusG,

pwBusGenericSensV, pwBusKVVolt, pwBusLambda, pwBusLoadMVA, // more comment

pwBusLoadMW, pwBusLongName])
```

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 `pwLineMW`. 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 `pwLineMW:0`, and for the to bus flow, we write `pwLineMW:1`. Note that fieldnames such as `pwLineMW:0` may simply leave off the `:0`.

## 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 `pwBusNum`, because a bus can be identified uniquely by its number. The key fields for GEN objects are `pwBusNum` and `pwGenID`. 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 main menu and choosing **Help > Export 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

### 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 possibly 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 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 red + 256\*green + 256\*256\*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

### 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:

Transmission Line or Transformer outage or insertion

```
BRANCH      | bus1# bus2# ckt | OPEN
                                     | CLOSE
```

Takes branch out of service, or puts it in service.

Series Capacitor Bypass or In Service

```
SERIESCAP   | bus1# bus2# ckt | BYPASS
                                     | INSERVICE
```

Interface outage or insertion

```
INTERFACE   | name | OPEN
                                     | CLOSE
```

Takes all monitored branches in the interface out of service, or puts them all in service

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.

Generator, Load, or Switched Shunt outage or insertion

```
GEN        | bus# id | OPEN
LOAD       | bus# id | CLOSE
SHUNT      | bus#
```

Takes a generator, load, or shunt out of service, or puts it in service.

Generator, Load or Switched Shunt movement to another bus

```
GEN        | bus1# | MOVE_PQ_TO | bus2# | value | MW
LOAD       |       | MOVE_P_TO  |       |       | PERCENT
SHUNT      |       | MOVE_Q_TO  |
```

Use to move generation, load or shunt at a bus1 over to bus2.

Generator, Load or Switched Shunt set or change a specific value

```
GEN        | bus# | SET_P_TO   | value | MW
LOAD       |       | SET_Q_TO   |       | PERCENT
SHUNT      |       | CHANGE_P_BY |       |
                                     | CHANGE_Q_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.

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 OPEN" "Interface L3+" CHECK //Drop ~1200 MW
</SUBDATA>
```

#### LimitViol

A LimitViol is used to describe the results of a contingency analysis run. Each Limit Violation lists four values:

**ViolType ViolElement Limit ViolValue**

<b>ViolType :</b>	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
<b>ViolElement :</b>	This field depends on the ViolType. for BAMP, BMVA – “bus1# bus2# ckt violationbus# for VLOW, VHIGH – “bus1# for INTER – “interfacename
<b>Limit :</b>	This is the numerical limit which was violated
<b>ViolValue :</b>	This is the numerical value of the violation

Example:

```
<SUBDATA LimitViol
BAMP "1 3 1 1" 271.94031 398.48096
INTER "Right-Top" 45.00000 85.84451
</SUBDATA>
```

#### 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 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 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 :**                    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: sgLineMW:0 = the MW flow leaving the from bus

SgLineMW:1 = the MW flow leaving the to bus

<b>Condition :</b>	Possible Values	Alternate1	Alternate2	Requirements
	between	><		requires othervalue
	notbetween	~><		requires othervalue
	equal	=	==	
	notequal	<>	~=	
	greaterthan	>		
	lessthan	<		
	greaterthanorequal	>=		
	lessthanorequal	<=		
	contains			
	notcontains			
	startswith			
	notstartswith			
	inrange			
	notinrange			

**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, [pwobjecttype, pwfiltername, pwfiltertype, pwprefilter])
{
  BUS "a bus filter" "AND" "no"
    <SUBDATA CONDITION>
      pwBusNomVolt > 100
      sgAreaNum  inrange "1 - 5 , 7 , 90-95"
      sgZoneNum  between
    </SUBDATA>

  BRANCH "a branch filter" "OR" "no"
    <subdata CONDITION>
      pwBusNomVolt:0 > 100 // Note location 0 means from bus
      pwBusNomVolt:1 > 100 // Note location 1 means to bus
      sgLineMW: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>
```

## 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.

### Example

```
//-----
// The following describes a multi-section line that connects bus
// 2 - 1 - 5 - 6 - 3
//-----
DATA (MultiSectionLine, [pwBusNum, pwBusName, pwBusNum:1, pwBusName:1,
      pwLineCircuit, pwMSLineNSections, pwMSLineStatus] )
{
2 "Two" 3 "Three" "&1" 2 "Closed"
  <SUBDATA Bus>
    1
    5
    6
  </SUBDATA>
}
```

## Data InjectionGroup

### PartPoint

A participation point is used to describe the contents of an injection group. Each participation point lists six values:

<b>PointType</b>	<b>PointBusNum</b>	<b>PointID</b>	<b>PointParFac</b>	<b>ParFacCalcType</b>	<b>ParFacNotDynamic</b>
<b>PointType :</b>					One of two values describing the type of violation GEN – a generator LOAD – a load
<b>PointBusNum :</b>					The bus number of the partpoint
<b>PointID :</b>					The generator or load id for the partpoint
<b>PointParFac :</b>					The participation factor for the point
<b>ParFacCalcType :</b>					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
<b>ParFacNotDynamic:</b>					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>
```

## 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.

**"AREA num1 num2** Monitor the sum of the tie line MW flows from area num1 to area num2.

**"ZONE num1 num2** Monitor the sum of the 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.

For the interface element type "BRANCH 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>
```

## 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 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 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.

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.

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.

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.

Example

```
<SUBDATA Branch>
  6 10 1 50 // shows 50% ownership of line from bus 6 to 10, circuit 1
</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, [pwDataGridName])
{
  BUS
  <SUBDATA COLUMNINFO>
    pwBusNomVolt 100 8 2
    sgAreaNum    50 8 2
    sgZoneNum    50 8 2
  </SUBDATA>

  BRANCHRUN
  <subdata COLUMNINFO >
    pwBusNomVolt:0 100 8 2
    pwBusNomVolt:1 100 8 2
    sgLineMW:0    100 9 3
  </SUBDATA>
}
```

## Data ATCScenario

### 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

**OTDF :** The OTDF on the limiting element under the limiting contingency

**LimitUsed :** The limit which was used to determine the MaxFlow

**PreTransEst :** The transfer

**MaxFlowAtLastIteration :** The total transfer at the last iteration

**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

**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.

**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 Schedule****LoadScale**

This subdata section contains a list of load schedule points. Each line of text contains two values, the time and the load multiplier.

Example

```
<SUBDATA LoadScale>  
  0 1.0  
 35 1.5  
 65 1.2  
</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
- PowerWorld Project initialization script
- Creating a new project file
- Opening an existing project
- Associating project files with simulator

## Overview

### 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:

- Open Windows Explorer.
- Select View, Folder Options from Explorers main menu.
- Switch to the tab labeled "File Types."
- Click the button labeled "New Type."
- For "Description of Type," specify "PowerWorld Project."
- For "Associated Extension," specify "PWP".
- For "ContentType (MIME)," specify "application/powerworld\_project."
- For "Default Extension," specify ".PWP".
- Under "Actions," click the button labeled "New."
- Specify the "Action" as "open."
- 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"

- Click OK to close the Action dialog.
- Click OK to complete the definition of the file association.

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 Project File

### 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 **File, Save As Project** from the main menu. 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 **File, Save As Project** from the main menu. 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 or in PTI Version 23 or 24-26 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.

#### Apply this template

Identifies a case template file that should be applied automatically when the project is open. Case templates store predefined sets of solution, environment, and display options. Including a template in a project ensures that the end user will see the Simulation exactly the way you see it on your computer. If your environment is set up to use a default template, the name of the default template will automatically appear in this text box. To specify a different template file, either type its full path in the text box, or press the adjacent *Browse* button to search for it on your machine.

**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 **Simulation, Play** from the main menu. 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 a Project File

### Opening an Existing Project

To open an existing PowerWorld Simulator Project file, select **File, Open Project** from the main menu, and select the name of the project you wish to open. Alternatively, you may select **File, Open Case** from the main menu, 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 and input the information it needs to display the case, oneline diagrams, and other associated files.

## Chapter 15: Web Publishing

Simulator presents several tools for saving a variety of data in a web-friendly format. You can save images as jpeg files, and all case information displays can be saved as HTML tables. You can then integrate these files into a single web document using Simulator's HTML editor. Finally, you can upload the HTML document, together with any necessary attachments, to your web server.

This chapter covers the following:

- Web Publishing Overview
- Saving Case Information Displays as HTML Tables
- Saving Images as JPEGs
- PowerWorld Simulator's HTML Editor
- The Publish to Web Dialog Box

### Web Publishing Overview

The best way to use the web publishing functionality is to follow these steps:

- Save all case information display contents you wish to publish on the web to separate HTML files. See *Saving Case Information Display Contents as HTML Tables* for details.
- Save all oneline diagrams and strip charts you wish to publish as jpeg files. See *Saving Images As Jpegs* for details.
- Launch the web publishing editor by selecting **File, Publish to Web** from the main menu. Use its tools to add content to either an existing document or a brand new document. You can insert the HTML code produced by saving the case information displays as HTML tables using the editor, and you can add links to the images you have saved as jpegs to the document, as well.
- Upload the main document, together with any attachments, to your web server using Simulator's built in ftp functionality.

### Saving HTML Pages in Simulator

#### 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 Display as HTML and when you try to insert a new table using Simulator's HTML editor.

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.

## **Editing HTML Pages**

### **HTML Editor**

To compose a web page using Simulator's HTML editor, select **File, Publish to Web** from the main menu. This opens the PowerWorld HTML Editor. The editor provides basic web page formatting functionality. It is not intended to compete with commercial packages that offer this as one of their primary functions, but it does give you tools for writing simple HTML pages that integrate the various resources you have produced using Simulator.

The web editor's window is divided into two sections. A set of command buttons occupies the left third of the display, and the HTML code for the document you are creating is shown in the text

editor occupying the rest of the window. Use the command buttons to add HTML tags to the document at the current cursor location. The command buttons are divided into several groups.

#### Title

Press the title button to specify a title for the web page. The title must be specified in the head section of the page.

#### Heading1 through Heading 5

Simulator's HTML editor allows you to apply formatting for five different heading styles. The actual appearance of each heading style depends on your browser's settings. The Heading 1 and Heading 3 styles are generally used to format main section and subsection headings, respectively.

#### Character Formatting

Use the **Bold**, *Italic*, Underline, Plain, Font Color, and Font Size buttons to apply various formatting styles to portions of your document's text.

#### Comment

Specify the comment text using the Comment command button.

#### Links and Images

To insert a hypertext link into the web document, click the **Link** button. This brings up the HTML Link Properties Dialog. To insert an image into the document, click the **Image** button. This brings up the HTML Image Properties Dialog. Fill out the requested information to add the link or image to the main document.

#### Page Spacing

Use the **Break**, **Paragraph**, **Rule**, and **Center** buttons to define the spacing of text in the document. **Break** inserts a hair line break; **paragraph** identifies a new paragraph, **Rule** inserts a straight line that spans the screen to separate different portions of the document, and **Center** centers the text horizontally on the screen.

#### List

The next three command buttons allow you to specify elements of either numbered or bulleted lists. Use the **List (1,2,3)** button to define a numbered list, and use the **List (\*)** button to define a bulleted list. This creates the list's skeleton. You can then fill the list skeleton with one or more list items by clicking the **Element** button.

#### Table

The four buttons in the Table section allow you to create a table from scratch. To define the table, click the **Table** command button. This brings up the Table Format Dialog, which allows you to specify the border weight, horizontal cell spacing, vertical cell spacing, table width, and caption. To have the browser employ its default settings in drawing the table, leave these options blank or set them to zero.

To insert a header row for the table, click the **Header** command button. This brings up the Table Format Dialog, which allows you to specify the vertical and horizontal positioning of the text in the header row.

To insert a row in the body of the table, click the **Row** command button. This brings up the Table Format Dialog, which allows you to specify the vertical and horizontal positioning of the text in the body row.

Finally, to insert data into the table, click **Cell**. This brings up the Table Format Dialog, which you can use to control the vertical and horizontal placement of text in the cell.

In defining the vertical and horizontal positioning of table text, cell-specific settings take precedence over row-specific settings. If no cell-specific settings are provided, however, those of the corresponding row are applied.

In addition to composing a web document from scratch, you can also load an existing document or insert text from an existing document into your new document. Choose **File, Load** from the editor's main menu to select an existing HTML document to load into the editor window. To insert code from another document at the current cursor location in the document you are creating (say, for example, to insert the html code for a table you created from a case information display), select **File, Insert** from the editor's main menu and select the file whose contents you wish to insert.

To save your new web document to your hard drive, click **File, Save** from the editor's main menu, and specify the file name you desire. To see how your document appears in your system's default browser, choose **File, Test** from the editor's main menu. (If you have not already done so, you will be asked to save the document locally before the test will begin.)

To publish the document on your web server, choose **File, Publish** from the editor's main menu. If you have not yet saved the document to your hard drive, you will first be asked to save it. After that, Simulator will open the Publish to Web Dialog. Use this dialog to set up and launch the transfer of the new document and its attachments (if any) to your web server.

To exit the web publishing editor, click **File, Close** from its main menu.

## HTML Link Properties Dialog

To insert a hypertext link in the web document, click the **Link** button on the HTML Editor. This brings up the HTML Link Properties Dialog, which sports the following fields:

### Descriptive Text

Specifies the text of the label on which the user will click.

### Web Address

Specifies the destination address of the link; that is, the address of the location to which the hypertext is linked.

### Upload This Local Copy

If the web page to which the document is linked resides on your local drive and you wish to send it to the web server along with the main document, check the Upload This Local Copy checkbox. Then, in the adjacent text box, specify the full local path of the linked document. Use the **Browse** button to find the document on your local drive if you would like. Also, specify whether the linked document is ASCII or Binary.

Click **OK** to finish inserting the hypertext link in the document you are composing. Click **Cancel** if you do not wish to add the link to the page.

## HTML Image Properties Dialog

To insert an image into the document, click the **Image** command button on HTML Editor. This brings up the HTML Image Properties Dialog, which requests the following information:

### Image Address

Specify the web address of the picture you want to display.

### Upload This Local Copy

Check this box to identify a file on your local PC that contains the image you are trying to insert. Provide the full local path of the image file in the adjacent text box. Use the **Browse** button to find the file on your local drive if you would like.

**Width, Height, Border Size, Vertical Spacing From Text, Horizontal Spacing From Text**

Use these controls to specify various display parameters for the image. Specify any value as zero if you would like to employ the browser's default setting for that option in rendering the image.

**Alignment**

Defines how the image will be aligned relative to the surrounding text.

Click **OK** to finish inserting the link to the image in the document you are composing. Click **Cancel** if you do not wish to add the image to the page.

## Publishing Pages to a Web Server

### Publish to Web Dialog

The Publish to Web Dialog allows you to send the document you have created using Simulator's HTML Editor, along with any attachments, to your web server. This dialog has two tabs, one labeled *Documents*, and the other labeled *Server*.

The **Documents Tab** identifies the document to upload and its attachments. It contains the following controls:

**Local Path**

Identifies the location of the document you have just written on the local drive.

**Full URL**

Specifies the full URL to assign to the page. The full URL is used to determine where on the web server to place the document. Therefore, be sure to include the base address of your web server in the Full URL specification. For example, if the page you have created is to be accessed from the PowerWorld website using the name mydoc.htm, specify the full URL path as `http://www.powerworld.com/mydoc.htm`.

**Attachments**

The Publish to Web Dialog provides a table identifying the other documents that are to be sent along with the main document. These other documents might include web pages that are linked to the main document you are uploading, or they might be images. To add an attachment to the attachments list, click the **Add New** button and fill out the Define New Attachment form. To delete an attachment from the list, click on the row of the attachment you wish to delete and click the delete key on your keyboard. To toggle the data type of an attachment listed in the Attachments table, double-click on the attachment's *Type* column. To toggle whether or not a specific column should be uploaded to the server, double click on the attachment's *Upload* column. To use the set of attachments associated with a previously published document, click the **Load Attachments From File** button. This will allow you to select a file identifying the attachments defined as part of a previous web publishing exercise. Such files are stored automatically with a *.met* extension whenever you upload a document to the web.

The **Server Tab** allows you to specify several parameters concerning how your web server is configured and accessed. The Server Tab hosts the following controls:

**User ID, Password**

Specify the user ID and corresponding password with which you access your web server's documents directory.

### Hostname

Indicate the machine name of your web server. Alternatively, you may specify the server's IP address.

### Port

Supply the number of the port on which the server will be listening for incoming documents. If you do not know this information, your best bet is to leave this field blank.

### Base URL and Corresponding Directory

The base URL is the base, or root, address of your website. This address corresponds to the root of the directory structure in which your site's web documents are stored, which is specified using the Corresponding Directory field. Be sure to specify the full path of the Corresponding Directory. The main document's full URL, the remote host's base URL, and the remote host's corresponding directory are used together to determine where to place the document you are uploading and its attachments. For example, suppose the new document's desired URL is `www.powerworld.com/cases/today.htm`. Furthermore, suppose the base URL for my web server is `www.powerworld.com`, and that the root directory for storing HTML documents on this server is `/my/web/directory`. Then, you should specify `www.powerworld.com` as the server's base URL, and `/my/web/directory` as the host's corresponding directory. As a result, the new document will be uploaded to the remote host that it will be placed in the remote host's `/my/web/directory/cases` directory. In other words, a new document will be created on the remote host called `/my/web/directory/cases/today.htm`. If any part of the remote path does not exist, Simulator will attempt to create it.

To upload the document to the server, click the **Upload** button. The text box in the lower right corner of the dialog will document what transpires during the communication with the remote server. When the file transfer is complete, a log identifying whether the main document and its attachments were transferred successfully and, if not, the reason for the failure.

To abort the web publishing activity, click **Cancel**. To close the dialog box at any time, click **Close**.

## Defining New Web Attachments

Web attachments are files that you want to send to the web server with the master document you have composed using Simulator's HTML Editor. Attachments might be jpeg images or other web pages to which the master document will link. The Publish to Web document lists your master document's attachments in a grid labeled *Attachments*. To add a new web attachment to this list, click the **Add New** button. This brings up the **Define New Attachment Dialog**, which requests the following entries:

### Local Path

The local path locates the document or file on your local drive. Use the **Browse** button to search for the attachment if you would like.

### Desired WWW Address

The address by which the attachment will be accessed over the world-wide web. Be sure to provide a complete URL in this box (including the server's base address) so that Simulator can determine in which directory on the host to place the attachment.

### Data Type

Identify the attachment's data as being either ASCII or binary.

Click **OK** to add the new attachment, or click **Cancel** to close the dialog without adding the attachment.

## Web Publishing Status

The **Web Publishing Status Dialog** documents whether the Simulator was able to transfer each of the files that comprise the web document you are publishing to your web server. Each line of the text box displayed on the **Web Publishing Status Dialog** lists the web address the document will have if it has been uploaded successfully, whether or not it was uploaded successfully, and, if not, the reason for the failure. Any document that was not uploaded successfully will likely appear as an invalid link to browsers of your site. To remedy this, try to address the problems (which may be related to network errors, bad directory file names on either the local or remote machine, or something else), and then try uploading the files again.



## Chapter 16: 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

#### Creating a New Case Page 1 of 10

This procedure describes how to create a simple power system model using PowerWorld Simulator. This procedure was developed for use with version 6.0 of the package. If you have an earlier version, please contact PowerWorld Corporation at [info@powerworld.com](mailto: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 the **File, New Case** from the main menu, or click the *Open Simulation Case* button on the File Palette. The screen background will turn yellow, the default background color for new PowerWorld online diagram. Online 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.

#### Inserting a Bus Page 2 of 10

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 online diagram, attach devices such as generators and loads to the buses, and connect different buses together with transmission lines and transformers.

Inserting buses on the online diagram is very easy:

- Select **Insert, Bus** from the main menu, or select the *Bus* button on the Insert Palette. This prepares Simulator to insert a new bus.
- Left-click on the online background at the location where you want to place the new bus. This invokes the Bus Option Dialog.
- Use the Bus Option Dialog Box to specify the name, size, orientation, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to it.
- 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 will appear on the online at the location you specified.

For example, let's add a bus to the new case. Select **Insert, Bus** from the main menu and click somewhere in the middle of the display. The Bus Options Dialog will open, inviting you to enter information about the bus. The *Bus Number* fields will automatically display '1'. Simulator requires that each bus have a unique number. For convenience, we will just accept the default value. In the *Bus Name* field, enter a name for the bus with up to eight characters. For this example, enter *ONE*. Next, check the System Slack Bus field, which is located toward the bottom

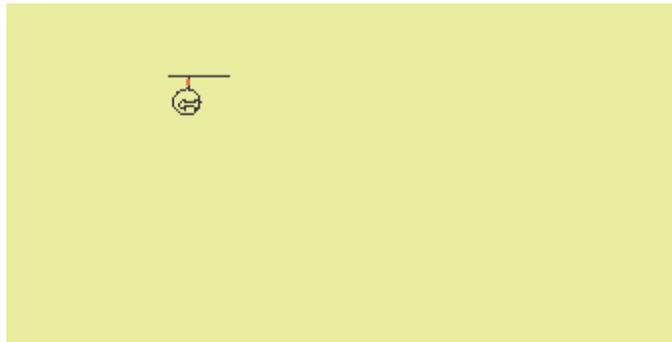
of the Bus Information Tab. The Slack Bus is a modeling construct that ensures that the power system has enough generation to meet the load. In other words, the slack bus “picks up the slack caused by system losses. For now, we will just leave the remainder of the fields with their default values. Click on the OK button at the bottom of the dialog to insert the bus. You should now see a single horizontal line in the middle of the oneline; this is your first bus.

## Inserting a Generator Page 3 of 10

Next we are going to attach a generator to the bus. Generators may be inserted by following a procedure that is very similar to the one used for inserting a bus:

- Select **Insert, Generator** from the main menu, or click the *Generator* button on the Insert Palette.
- Left-click the bus on the oneline diagram to which you want to attach the generator. This brings up the Generator Option Dialog.
- Using the Generator Option Dialog, specify the new generator's unit identifier, display size, orientation, MW output and limits, reactive power limits, setpoint voltage, and cost model.
- Click *OK* on the Generator Option Dialog to finish creating the generator and to close the dialog. After the dialog box closes, the new generator will appear on the oneline attached to the bus on which you clicked in step 2.

Follow this procedure to attach a generator to the slack bus, bus 1. When the generator option dialog box appears, simply accept the default fields. The oneline diagram should resemble the one shown below.



## Saving the Case Page 4 of 10

To save the work that we have done so far, select **File, Save Case** from the main menu. 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, simply click the Log button on the Program Palette. Since we have not yet named the case, the Save As dialog is displayed. Enter the filename 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.

## Entering a Second Bus with Load Page 5 of 10

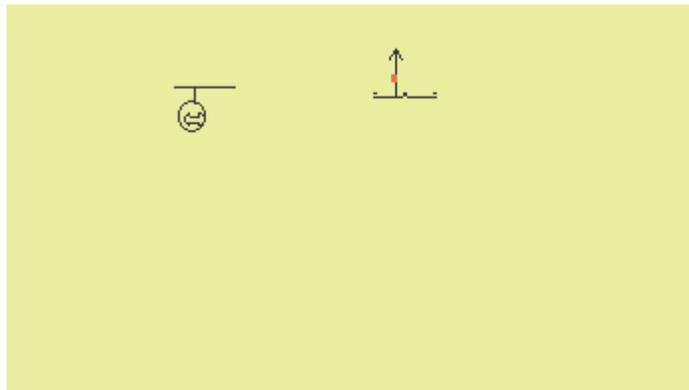
To enter the second bus, select **Insert, Bus** from the main menu or click the *Bus* button on the Insert Palette. Then click on the oneline diagram somewhere to the right of the first bus. The Bus Options Dialog will open. Leave the bus number at the default value of 2, and enter the name *TWO* for the bus. We also want to model a 200 MW, 100 MVR load at the bus. To accomplish this, switch to *Attached Devices* tab and, under the *Load Summary Information* heading, enter 200 in the *Base MW* field and 100 in the *Base Mvar* field. Click on the OK button at the bottom of the dialog to insert the bus.

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 **Insert, Load** from the main menu, or select the *Load* button on the Insert Palette. Now left-click the mouse somewhere towards the right edge of this bus. You should immediately see the Load Options dialog box, whose Constant Power MW and MVR fields confirm that the load is 200 MW and 100 MVR. In addition to constant power loads, Simulator also allows the modeling of voltage dependent load.

The Orientation field determines the orientation of the symbol when it is drawn on the oneline. Check the Up entry to make the load point up. Also, verify that the anchored box is checked. By anchoring the load to the bus, the load moves anytime you move the bus. Then click on the OK button to insert the load. A circuit breaker symbol is automatically included with each load.

Now, try moving bus 2. To do this, click somewhere on the bus (not on the load). Holding down the left mouse, move the bus (this is known as “dragging”). The load should move as well, because it is anchored. To reposition the load on the bus, simply click on the load, and then drag it (holding the left button down) to a new position on the bus.

The oneline diagram should now resemble the one shown below.



## Inserting a Transmission Line Page 6 of 10

To connect the two buses together, we will now insert a transmission line between them. The procedure for inserting a transmission line involves the following steps:

- Select **Insert, Transmission Line** from the main menu, or click the *Transmission Line* button on the Insert Palette.
- 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.
- Without holding down the mouse button, drag the mouse. You will notice that a line segment connected to the point of origin will follow your mouse movements. Transmission lines and

transformers are drawn as a series of line segments. To terminate a line segment, simply click the left mouse button. To begin drawing the next line segment, simply move the mouse again. Each time you click the mouse to terminate a line segment, a new vertex is defined for the line. These vertices may later be moved or deleted to reshape the line.

- To terminate the final line segment comprising the line (and thus conclude drawing the line), double click the left mouse button at the point where the line should terminate. The termination point is usually the line's other terminal bus. The Transmission Line/Transformer Dialog will appear.
- Specify the new line's parameters using the Transmission Line Dialog and click *OK*. The new transmission line will appear on the oneline.

We will now insert a transmission line between buses 1 and 2. Select **Insert, Transmission Line** from the main menu, and click at the point on bus 1 at which you would like the line to start. This begins the process of inserting the line. Move the cursor upwards a short distance and then click the left mouse button to terminate the first segment. Then move the mouse horizontally until the cursor is above bus two and click the left mouse button to define the second segment. Finally, move the cursor down to the location on bus 2 where you would like to attach the line and double-click. You should immediately see the Transmission Line/Transformer Dialog. The dialog should already contain a 1 in the *From Bus Number* field and a 2 in the *To Bus Number* Field. The *Resistance*, *Reactance*, and *Capacitance* fields are used to enter the per unit parameters associated with the line. The Capacitance field contains the total per unit charging capacitance for the line. Enter a 0.02 in the Resistance field, 0.08 in the Reactance field and 0.1 in the Capacitance field. The MVA Rating field contains the MVA rating for the line; enter a value of 400.

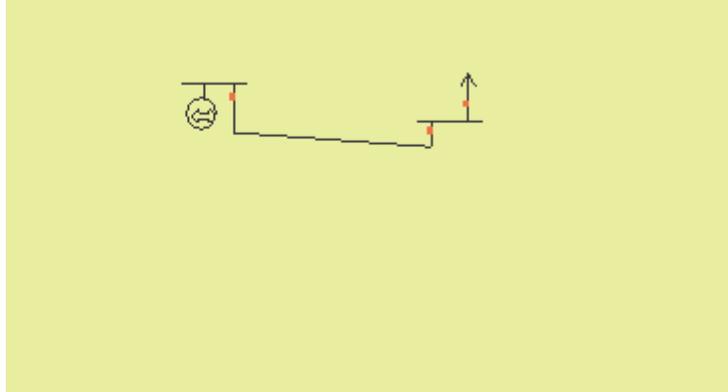
If the dialog did not contain a '1' in the From Bus Number field and/or a '2' in the To Bus Number field, you probably did not have the cursor directly on the bus when you were drawing the line. Simply enter the correct bus numbers.

Finally, click the OK button to insert the transmission 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.

We will now place circuit breakers on 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). Circuit breakers are used to control the status of the line. Click somewhere on the line near bus 1. Then select **Insert, Circuit Breaker**. You should immediately see the Circuit Breaker Options dialog 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 in change the value). Click the OK button 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, just repeat the above process near bus 2.

**Save your case.** Your oneline should now look similar to the one below.



## Inserting Text, Bus and Line Fields Page 7 of 10

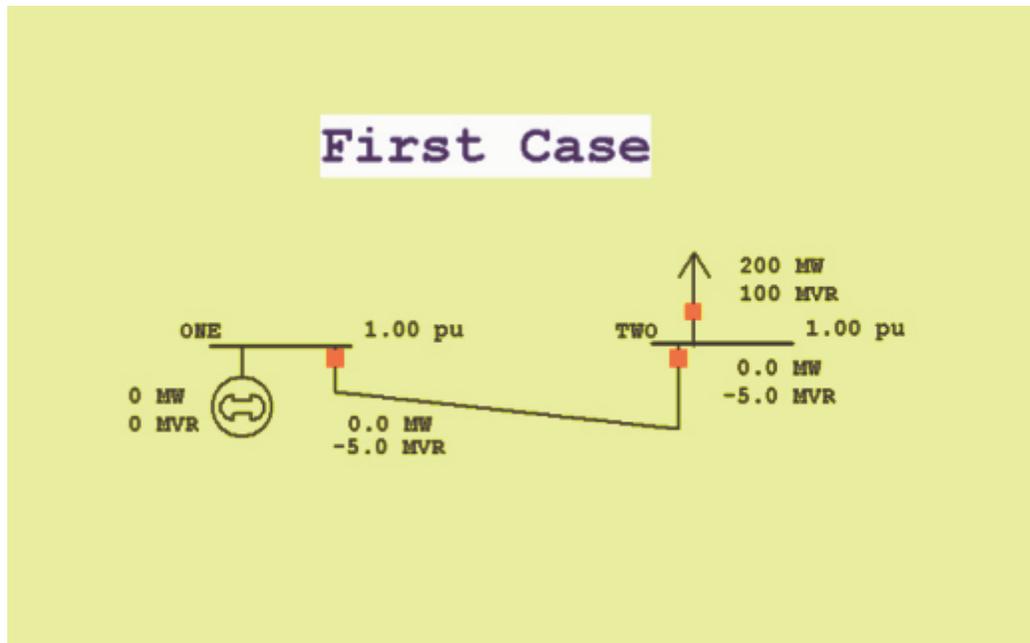
We are almost ready to run our FIRST case. However, before we do that, we are going to enter a few informational fields directly on the oneline so that we can see what is going on in the system more easily. First, we will add a title to the diagram. Select **Insert, Text** from the main menu to bring up the *Text Object Dialog*. Type the string “First Case and click *OK*. To make the text look more like a title, we will do some quick formatting. Select **Format, Font** to change the font. This displays the Font Tab of the Format Selection Dialog. Make the font size 26, and change the font color blue. To change the text background color, select **Format, Line/Fill**, which summons 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*. Finally, if you would like to move the text, simply left click on it and drag it to the new location.

Next, we will add several fields that display bus-related quantities. To add fields to the display of a particular bus, follow this procedure:

- 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.
- Use the Insert Bus Fields Dialog to designate the fields to add. You may add up to 8 fields per bus.
- Click *OK*. The specified bus fields will be added to the oneline diagram.

For this example, we will add a Name, Bus Voltage Magnitude, Actual Gen MW, and Actual Gen MVR fields to bus 1, and Name, Bus Voltage, Load MW, and Load MVR fields to Bus 2. We can accomplish this in four steps. First, right click on bus 1 and select the *Add New Fields Around Bus* option, which brings up the Insert Fields dialog. Because Bus 1 is horizontal, the possible positions for the fields we will be adding are shown in the diagram located in the top right half of the dialog. We will insert the Name field for bus 1 in position 1 and the voltage field in position 5. Use the combo boxes to specify assign the *Name* field to position 1 and the *Voltage* field to position 5, and press *OK*. The name and voltage fields are to the oneline diagram around bus 1 in the positions we specified. Repeat this same process for bus 2. If necessary, you may move each of the four newly added fields manually with the mouse. Next add the Actual Gen MW and Actual Gen MVR fields to the generator at bus 1. Right-click on the generator and select the *Add New Fields Around Gen* option to bring up the Insert Fields dialog. Since the generator is oriented down, the possible positions for the new fields we are adding are shown in the rightmost illustration on the top half of the dialog. Use the dropdown boxes to assign Actual MW to position 1 and Actual MVR to position 2. Finally, let us add the MW and MVR fields for the load at bus 2. Right-click on the load and select the *Add New Fields Around Load* option to invoke the Insert Fields dialog. Since the bus is oriented upward, the relevant position diagram is the second one from the left. Use the dropdown boxes to assign MW to position 5 and Mvar to position 6.

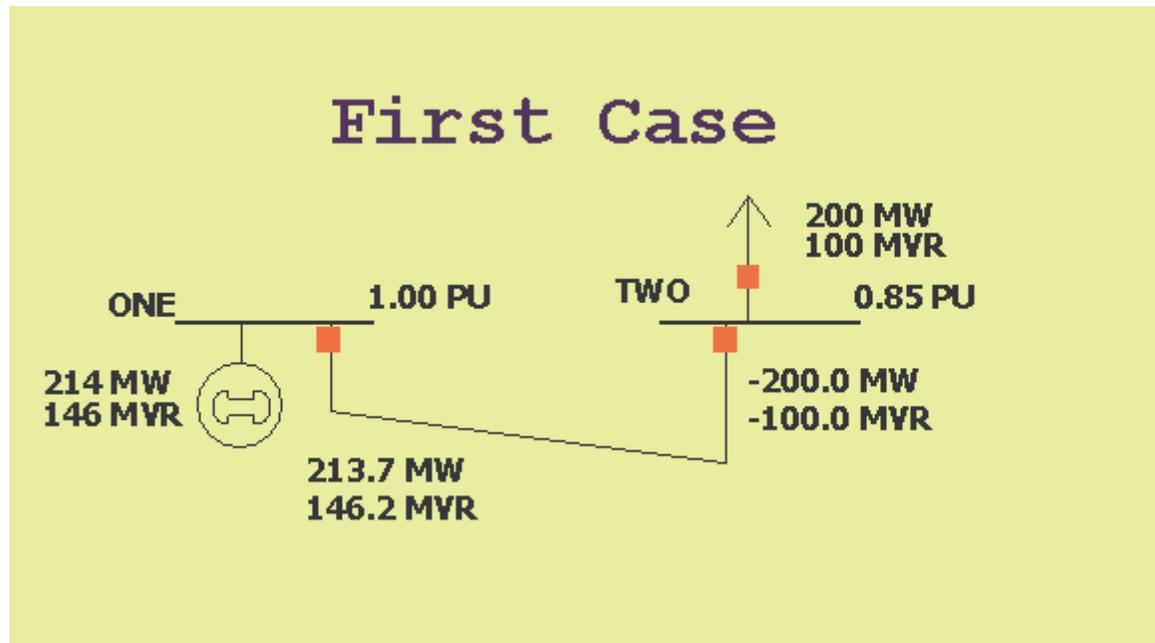
Next we will insert fields showing the power flow on the transmission line. Right click on the line between buses 1 and 2 and select the *Add New Fields Around Line* option to bring up the Insert Fields Dialog. Use the dropdown boxes to assign MW Flow to position 3 and Mvar Flow to position 4, and MW Flow to position 9 and Mvar Flow to position 10. Click *OK* to add the fields to the oneline. You may need to move the newly inserted fields with the mouse to improve their appearance. At this point, your first oneline diagram should resemble the one shown below.



## Solving the Case Page 8 of 10

We are now ready to solve the case. First we need to enter the Run Mode. To do this, simply click on the Run Mode button on the Program Palette. Note that, if the case has validation errors, you will need to rectify the problems before you can enter Run Mode. To start simulating the case, simply select **Simulation, Play**. Alternatively, to perform just a single Power Flow Solution, click *Single Solution* on the Program Palette. 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. If the Log window is visible, you will get a "backstage view of what Simulator is doing. Feel free to close the log. Then, with the load circuit breaker closed, try clicking on a transmission line circuit breaker.

Congratulations, you've just blacked-out your case!



### Adding a New Area Page 9 of 10

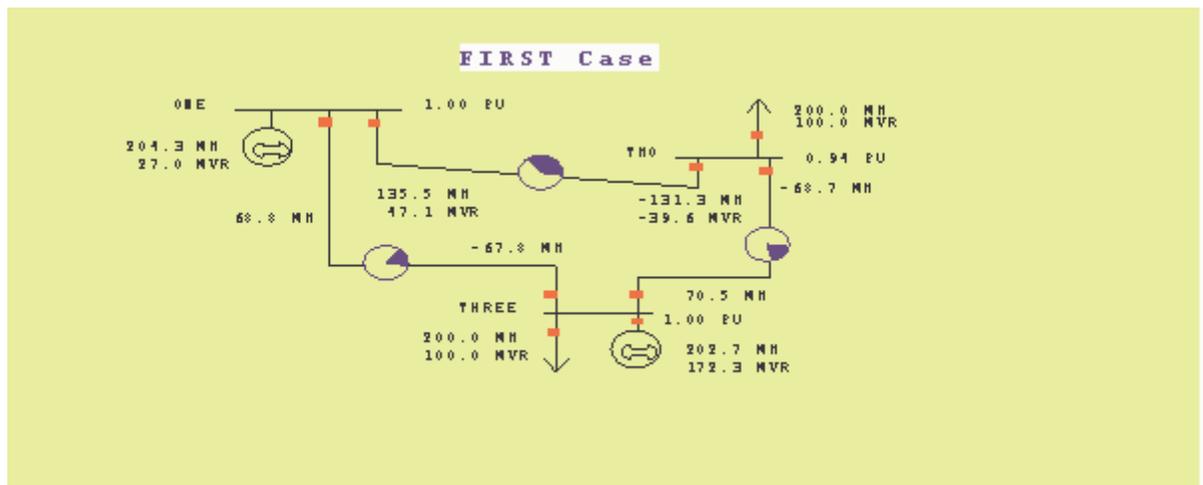
Next, we will create a second operating area for the case. Large interconnected systems usually have a number of control areas, with each control area responsible for the operation of a particular part of the system. Often, a single control area corresponds to a single owner (such as an investor-owned 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, go back to editing by clicking the Edit Mode button. Then add a third bus located below buses one and two. Set the bus number to '3' and the bus name to 'THREE'. Set the Base MW field to 200 and the Base Mvar field to 100. However, in the Area Number field enter '2'; in the Area Name field enter 'TWO'. Entering a number for an area that does not already exist automatically creates a new area. Next, enter a generator and a load at bus three. Then enter a transmission line between buses 1 and 3 and another between buses 2 and 3. For each line enter a 0.02 in the Resistance field, 0.08 in the Reactance field, 0.1 in the Capacitance field, and 400 in the MVA Rating field.

To verify that the case now has two areas, select **Case Information, Case Summary**. The Case Information displays allow you to view the entire case using non-graphical displays. The Case Summary dialog shows the number of buses, generators, lines/transformers, and control areas in the case. You cannot modify any of these values. 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, Areas**. The Area display provides a convenient summary of all the control areas in the case. Similar displays exist for buses, generators, lines/transformers, etc. Right-click anywhere on the record for the first area and select the *Show Dialog* option. The Area Display Dialog is shown. Change the Area Name to 'ONE' and set the AGC Status to ED 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 ED Control, and select OK to save your changes and close the dialog. Close the Area Records display as well.

Next, we will insert line flow pie charts on each transmission line. The line flow pie charts are used to show the percentage MVA loading of a line or transformer graphically. Select **Insert, Line Flow Pie Chart** from the main menu, or click the *Line Flow Pie Chart* button on the Insert Palette. Click near the middle of the line from 1 to 2. The *Line/Transformer Flow Pie Chart* will appear, with the From, To, and Circuit fields set automatically to identify the line from bus 1 to bus 2. Click *OK* and observe that a pie chart is added to the diagram on the line between buses 1 and 2. Repeat this process for lines 1-3 and 2-3. Then insert text, bus, and line fields, and finally solve the case using the Run Mode. Your case should look similar to the case shown below. While the simulation is running, click on the circuit breakers and note the nearly instantaneous change in system flows.



## Case Customization Page 10 of 10

Simulator allows you to customize your case in a number of different ways. We will touch on just two of these options in this section.

For example, suppose we want to prescribe how the load should vary over the course of the simulation. To accomplish this, first switch back to Edit Mode, and then select **Case Information, Others, Load Variation Schedules** from the main menu. The Load Variation Schedule Dialog will appear. Use the Load Variation Schedule Dialog to define how load should vary over time. These prescriptions are stored as records which may then be associated with a particular area or areas to govern how the load changes in those areas during the simulation. At this time, no load schedules exist, and you will thus have to define one. Designate the *Schedule Type* as *Daily*. Then, edit the table that lists time points and corresponding load scales such that the load scale is 1.000 at hour 0, 0.8 at 06:00, 1.3 at 12:00, 2.0 at 18:00, and back down to 1.000 at hour 24:00. This load schedule might approximate a control area's typical summer load variation, with peak load occurring in the late afternoon. Then click the *Save Record As* button to save the schedule with name *Area1Daily*. Then, to associate the *Area1Daily* load schedule with Area 1, choose **Case Information, Areas** from the main menu. Right-click on the record corresponding to area 1 and select the *Show Dialog* option. In the resulting *Area Information Dialog*, designate the load schedule as *Area1Daily*. Then, when you restart the simulation, you will find that load in area 1 will vary according to the *Area1Daily* load schedule. The same load schedule can be applied to the other area in the case, or you may define a new load schedule.

As another example of customization, we will explore how to change the animation of power flows. Select the *Animated Flows Options Tab* of the *Online Display Options Dialog* by

selecting **Options/Tools, Online Display** from the main menu. Check the *Show Animated Flows* box and the *Use Fill Color* box. Then select the Change Fill Color button. Select a greenish color. Then choose *OK* to close the Color dialog and *OK* again to close the Online Display Options Dialog. Save the case and again start the simulation in the Run Mode. You should see the power flowing down the line. Notice that the load in area 1 is changing and that the generation changes as well to maintain the balance between supply and demand.

## Starting with an Existing Case

### 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 procedure was developed for use with version 6.0 of the package. If you have an earlier version, please contact PowerWorld Corporation at [info@powerworld.com](mailto:info@powerworld.com) for information on upgrading, or visit the website at <http://www.powerworld.com>.

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 easily be created from existing power flow cases stored in PTI version 24, PTI version 23, and IEEE common format. Simulator provides a static model of a power system. For some studies this model is sufficient. For other studies the model can be augmented by adjusting generator cost information, the reactive capability curve, the time variation of the load, and the PowerWorld case options.

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.

### 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 on the Program Palette. Select the **File, Open Case** from the main menu, or select the **Open Case** button on the File Palette. 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 yellow. 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.

### 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 Information, Case Description** from the main menu. 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 Information, Case Summary** from the main menu. 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 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 **Case Information, Area/Zone Filters** from the main menu, or use the **Area/Zone Filters** button on the Options/Info Palette. 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 **Case Information, Power Flow List**. 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 **Case Information, Quick Power Flow List** from the main menu, or press the **Quick Power Flow List** button on the Options/Info Palette. 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.

## Solving the Case Page 4 of 15

To solve the power flow case we have been using in this example, press the Single Solution button on the Program Palette. 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.

## 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, for 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.

## 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 **Insert, Bus** from the main menu, or click the **Bus** button the Insert Palette. 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 **Insert, Field, Bus** from the main menu. Alternatively, click the **Bus Field** button on the Insert Palette. 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.

## Automatic Line Insertion Page 7 of 15

Transmission lines between buses can be inserted manually by choosing **Insert, Transmission Line** from the main menu or pressing the **Transmission Line** button on the Insert Palette, clicking on the beginning bus, and tracing a line to the ending 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 **Insert, Auto Insert, Lines** from the main menu in Edit Mode. 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. Alternatively, use the Pan/Zoom Palette. For the example, pan up and insert buses 39819 and 39841, again with bus analogs showing their numbers. Again use **Insert, Auto Insert, Lines** to add the transmission lines.

## 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 **Insert, Background Graphic, Background Line**, or click the **Background Line** button on the Insert Palette. Click on the diagram to start the background line and 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 **Format, Line/Fill** from the main menu 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.

## 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 cannot only show the magnitude and direction of flows on transmission lines, but it can also *animate* them. To configure the animated flows, switch to Edit Mode and choose **Options/Tools, Online Display** from the main menu. Select the *Animated Flows Tab* of the resulting dialog box. Check the **Show Animated Flows** check box to enable the animated flows at run time. 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 **File, Save** from the main menu or by pressing the **Save** button on the File Palette. 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 perform the simulation, switch to Run Mode and select **Simulation, Play** or **Simulation, Restart** from the main menu. You should now see Simulator modeling the 3990-bus case.

## Run-time Object Dialogs Page 11 of 15

While the simulation ensues, you might want to view or change some study parameters. To do this, first pause the simulation so that you will not lose any simulation time while you are viewing or adjusting the parameters. (Note that you do not *have* to pause the simulation to tweak parameters. You can adjust anything in the case while the simulation runs, too.) Then, right-click on any of the objects on the display. 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 simulation 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 to display. 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.

## Area Records Page 13 of 15

Often, system data is most conveniently displayed by area. To view the Area Records Display, select **Case Information, Areas** from the main menu. 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 **Case Information, Limit Violations** from the main menu. A display showing bus voltage violations, line/transformer violations and interface violations will appear. 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 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 oneline. Simulator's interface has been designed to be simple and intuitive. If you run into problems, the On-line Help should prove helpful.



# Index

## A

AAC Files .....	42
ACE .....	241, 242
Strip Chart .....	246
Adding display objects to a oneline .....	53, 54
ac Transmission Lines .....	63
Area Fields .....	57
Area/Zone.....	56
Background Shapes.....	75
Bus Fields.....	60, 61
Buses .....	59, 60
DC Transmission Lines .....	65
Generators.....	61, 62
Interface Fields .....	72
Interfaces.....	70, 71
Line Fields .....	64
Loads .....	62, 63
Series Capacitor .....	68, 69
Switched Shunts .....	69, 70
Text .....	76
Transformer Fields.....	67
Transformers .....	66, 67
Admittance Matrix.....	50
Advanced Filtering.....	177
Advanced Filters Dialog.....	177
Advanced Filters Display .....	178
Advanced LODF Calculation Dialog.....	291
All Area Gen IC Curves .....	252
ALT Key .....	103
Used to temporarily disable snap-to-grid .....	103
Anchored Objects.....	55
Animated Flows Options.....	28
Animation.....	30
Contours.....	281
Showing and Customizing Power Flows .....	25
Appending Cases .....	96
Area Control.....	241, 242
Area Control Error (ACE).....	241, 242
Area Control Error Chart.....	246
Area Fields	
Edit Mode Dialog.....	160
Representation on Onelines.....	57
Area Information Dialog .....	156
Information.....	157
<b>Interchange</b> .....	157, 158
<b>Options</b> .....	158
Scheduled MW Transactions.....	160
Simulation Summary.....	159
Tie Lines.....	159
Area Interchange Tolerance .....	157
Area Records .....	190

<b>Area Slack Bus Control</b> .....	241
Area Transaction Options Dialog .....	244
Area/Zone Load Schedule Format (*.shd).....	41
Areas	
Area/Zone Display Options .....	56
Area/Zone Filters .....	176
Average Cost Chart .....	247
Fields on Onelines .....	57
Finding on the Onelines .....	103
Information Dialog.....	156, 157
Load and Generation Chart.....	246
Load Schedules .....	190
Load Variation Schedules .....	164, 201, 202
Losses Chart.....	246
Marginal Losses for a Transaction.....	285
MW Transactions Chart .....	247
OPF Records.....	323
Records .....	190
Representing on onelines.....	56
Scaling Load	
Generation and Shunts.....	98
Sorting.....	190
Super Areas.....	162
Arrow Keys	
Used to Pan Displays.....	31
ATC	
ATC Extra Monitors Dialog.....	337
Available Transfer Capability (ATC) Analysis.....	333
Available Transfer Capability Dialog.....	334
Multiple Scenario Available Transfer Capability Dialog .....	338
Transfer Limiters Display.....	340
Attachments	
Adding to a web page .....	382
Auto Insert Directions .....	292
Automatic Area Controls.....	41
Automatic Area Controls Format (*.aac).....	42
Automatic Generation Control.....	240
Automatic Line Tap Dialog.....	82
Automatic Voltage Regulation	
Generators.....	120
Transformers .....	135
Automatically Defining Interface Records.....	71
Automatically Inserting Borders .....	95
Automatically Inserting Buses .....	90
Automatically Inserting Generators .....	92
Automatically Inserting Interfaces .....	93
Automatically Inserting Lines and Transformers.....	90
Automatically Inserting Loads .....	92
Automatically Inserting Substations .....	94
Automatically Inserting Switched Shunts.....	93
AUX Files .....	41, 42
Auxiliary Data File Type.....	43
Auxiliary File Format (*.aux).....	42
<b>Auxiliary Files</b> .....	38, 41
*.aac Format .....	42
*.aux Format .....	42

*.gcd Format .....	44
*.gcp Format .....	44
*.inf Format .....	46
*.inj Format .....	46
*.shd Format .....	41
Available Transfer Capability	
Available Transfer Capability (ATC) Analysis.....	333
Available Transfer Capability Dialog .....	334
Multiple Scenario Available Transfer Capability Dialog .....	338
Transfer Limiters Display.....	340
AVR	
Generators.....	120
Transformers .....	135
<b>B</b>	
Background Color.....	104
Changing .....	104
Background Ellipses .....	75
Background Lines.....	74
Background Pictures .....	75
Background Rectangles.....	75
Base MW Transactions Display .....	211
Base Transactions	
Modeling.....	243
Bitmaps	
Exporting Onelines As.....	49
Blackouts .....	240
Borders .....	95
Automatically Inserting.....	95
<b>Boundary Buses</b> .....	96
Building a new oneline diagram .....	40
Building a power flow case .....	40
Bus Admittance Matrix.....	50
Bus Display Objects .....	59
Bus Fields .....	55
Edit Mode Dialog.....	111
Representation on Onelines .....	60
Bus Information Dialog .....	142
Bus Records.....	194
Bus Renumbering	
Automatic Setup of Bus List Options .....	78
Bus Change Options .....	79
Bus Renumbering Dialog.....	77
Bus Selection Dialog .....	41, 86, 96
Bus View .....	247, 248
Buses.....	90
Automatically Inserting.....	90
Case Information Display .....	194
Changing Zones .....	109
Contouring.....	281
Edit Mode Dialog.....	109
Fields on Onelines .....	55, 60
Finding on the Onelines .....	103
Information Dialog.....	142
Inserting.....	59
Inserting and Placing Multiple Fields .....	55

Losses .....	293
Mismatches .....	196
OPF Records .....	320
Quickly Showing Incident Flows .....	188
Records .....	194
Remotely Regulated.....	194, 195
Representation on Onelines .....	59
Scaling Load	
Generation and Shunts.....	98
Sorting .....	194
Splitting Buses .....	79

**C**

Case Description .....	185
Case Formats.....	37, 40
Case Information Display Options.....	234
Case Information Displays	
Advanced Filters .....	177
Area Records.....	190
Area/Zone Filters .....	176
Bus Records .....	194
Cell Handles .....	175
Colors and Cell Styles.....	175
Copying and Pasting Data.....	181
DC Line Records.....	207
Defining What to Show.....	171
Description.....	185, 186
Directions Display .....	292
Find Dialog .....	176
Formatting .....	171
Generator Cost Records .....	199
Generator Records .....	197
Interface Records.....	209
Limit Violations List .....	214
Line/Transformer Records.....	202
Load Records .....	201
Local Menu .....	173, 174
Make Reports .....	189
Mismatches .....	196
Multi-Section Lines.....	204
MW Transactions .....	211, 212
Options Dialog .....	171
Overview .....	171
Power Flow List .....	187, 188
Quick Power Flow List.....	188, 189
Remotely Regulated Buses.....	194
Saving as HTML .....	377
Saving Data .....	182
Sorting.....	176
Summary .....	186
Super Area Records.....	192
Switched Shunt Records.....	208
Transformer Impedance Correction Records.....	135
Transformer Records .....	205
Zone Records .....	191
Case Summary.....	186

---

Case Templates .....	51
Case Validation .....	38
Cases	
Appending .....	77, 96
Opening .....	39
Recently Opened .....	40
Saving .....	40
Scaling.....	98
Total Load	
Generation	
Shunts and Losses.....	186
Cells	
Colors and styles for case information displays .....	175
Handles .....	175
Changing.....	104
<b>Background Color</b> .....	105
Default Font.....	104
Display Appearance.....	104
Font.....	105
<b>Line/Fill</b> .....	105
Circuit Breakers.....	64
Close Online.....	41
Colors	
Background .....	105
KV-Based Coding .....	85
Conditional Display.....	34
Context Sensitive Help .....	17
Contingency Analysis	
An Introduction.....	255
Auto Insert Contingencies.....	273
Comparing Contingencies List Displays.....	267
Comparing Contingency Analysis Results .....	266
Contingencies Tab .....	260, 261
Contingency Analysis Dialog.....	255
Contingency Case Dialog.....	256
Contingency Case References.....	275
Contingency Definition Display.....	262
Contingency Element Dialog.....	257
Contingency Make-Up Power Sources Dialog .....	259
Contingency Solution Options Dialog .....	257
Contingency Violations Display .....	263
Lines Buses and Interfaces Tab .....	264
Loading Contingencies From a File .....	274
Make-Up Power Sources .....	258
Options Tab .....	270
Options Tab Limit Monitoring.....	271
Options Tab Miscellaneous .....	273
Options Tab Modeling.....	270
Options Tab Report Writing.....	272
Other Contingency Actions.....	265
PSS/E Contingency Format.....	259
Running the Contingency Analysis .....	259
Saving Contingency Records to a File.....	275
Summary Tab .....	273
Continue.....	239
<b>Continuing</b> .....	239

Contour Type .....	281
Contour Type Options .....	283
Contouring	
Options .....	281
Overview .....	281
Contouring Options	
Contour Type .....	281
Contour Type Options .....	283
Custom Color Map .....	284
Control .....	239, 240
Coordinates .....	31
Copy Command .....	107
Copying Data To/From Other Applications .....	181
Copying Objects .....	107
Copying Oneline Diagrams .....	31
Cost	
Average Area Cost Chart .....	247
Cubic Function Coefficients .....	118
Cost Modeling .....	118, 252
Auxiliary Files .....	41
Cost Records .....	199
Create Project Dialog .....	375
Creating	
Oneline diagram .....	40
Power Flow Case .....	40
CTRL Key	
Used for adding/deleting line vertices .....	63, 65, 74
Used with Arrow Keys to Zoom .....	31
Custom Color Map .....	284
Custom Expressions .....	178
Custom Expressions Display .....	179
Customizable Popup Hints .....	237
Customized Data Grid Display .....	183
Cut Command .....	107

## D

DC Transmission Line Information Dialog	
Actual Flows .....	130
Inverter Parameters .....	130
Line Parameters .....	128
Rectifier Parameters .....	129
DC Transmission Line Records .....	207
DC Transmission Lines	
Case Information Display .....	207
Information Dialog .....	128
Inserting .....	65
Records .....	207
Representation on Onelines .....	65
Sorting .....	207
Default	
Background Color for Onelines .....	103
Drawing Options .....	83, 104
Sizes of Objects .....	83
Default Font .....	106
Changing .....	104
Define New Attachment Dialog .....	382

Delete Command .....	108
Deleting Objects .....	107, 108
Difference Flows.....	251, 252
Directions Dialog .....	291
Directions Display.....	292
Display Appearance Dialog.....	104
Display Objects	
Adding to a Oneline .....	53
Area Fields .....	57
Areas .....	56
Background Ellipses.....	75
Background Lines .....	74
Background Rectangles .....	75
Bus Fields.....	60
Buses .....	59
Circuit Breakers .....	64
DC Transmission Lines .....	65
Generators.....	61
Interface Fields .....	72
Interfaces.....	70
Line Fields .....	64
Line Flow Pie Charts.....	65
Loads .....	62
Modifying Appearance .....	53
Moving around objects on a oneline .....	53
Relationship with Model Data .....	23
Resizing Objects.....	53
Series Capacitor Fields.....	69
Series Capacitors.....	68, 69
Switched Shunts .....	69
Text .....	76
Transformer Fields.....	67
Transformers .....	66
Transmission Lines .....	63
Zones .....	56
Display Options .....	26
Distribution Factors	
Line Outage Distribution Factors	
Line Outage Distribution Factors (LODFs) .....	288
Line Outage Distribution Factors (LODFs).....	288
LODF	
Line Outage Distribution Factors (LODFs) .....	288
Power Transfer Distribution Factors .....	285
Document Links.....	74
Drawing Coordinates .....	31
Drawing Grid Dialog .....	103
Drawing Options.....	83
<b>E</b>	
<b>Economic Dispatch Control</b> .....	241
Loss Sensitivities .....	293
Edit Menu	
Copy Command.....	107
Cut Command.....	107
Delete Command.....	108
Overview .....	101

Paste Command .....	108
Edit Mode	
Area Fields Dialog.....	160
Bus Dialog .....	109
Bus Fields Dialog .....	111
General Procedures .....	53
Generator Dialog.....	114
Interface Dialog.....	165, 166
Introduction.....	18
Line Fields Dialog .....	127
Line/Transformer Dialog.....	123
Load Dialog .....	120
Overview .....	53
Switched Shunts Dialog .....	140
Edit Palette.....	21
Enforcing Area Interchange .....	157
Equivalents	
Case Construction .....	96
Overview .....	96
Equivalents Display .....	96
Exporting Onelines .....	49
<b>External System</b> .....	96

## F

F1 Key (Press for Help) .....	17
Fault Analysis	
Bus Records .....	279
Dialog.....	276, 277, 278
Fault Analysis Dialog.....	276
Generator Records .....	279
Line Records.....	279
Mutual Impedance Record Dialog .....	280
Fault Analysis Dialog .....	276
Fault Analysis:.....	276, 279
Fields	
Case Information Displays .....	171, 173
File Menu .....	37
File Palette .....	21, 37
File Types .....	37, 40
Filters	
Advanced Filters .....	177
Area/Zone Filters .....	176, 177
Find	
Areas on Onelines .....	103
Buses on Onelines.....	103
Records on Case Information Displays.....	176
Find Dialog Basics.....	179
Flat Start .....	238, 240
Flowgates	
Reading/Writing NERC Format .....	73
Font	
Changing .....	104
Properties .....	106
Format Menu.....	104, 105
Display Appearance.....	107
Display Background Color.....	103

Display Drawing Grid .....	103
Display Zoom/Pan.....	103
Line/Fill Options .....	106
Format Palette.....	21
Format Selection Dialog .....	105
Font Tab.....	106
Line/Fill Tab.....	106
Object Display Properties.....	107
Frequency	
Constant Frequency Model .....	240
Fuel Cost Curve .....	252
Functions and Operators Available .....	183

## G

GCD Files .....	41, 44
GCP Files.....	41, 44
Generation Shift Factors.....	295
Generator Capability Curves Format (*.gcp).....	44
Generator Cost Curve Data Options .....	45
Generator Cost Data Format (*.gcd) .....	44
Generator Cost Records.....	199
Generator Fields.....	55
Generator Information Dialog.....	118
Cost Data.....	115
Display Information .....	115
MW Control.....	116
Short Circuit Parameters .....	118
Voltage Control .....	117
Generator Participation Factors .....	119
Generator Records.....	197
Generator Run Mode Dialog	
Input-Output Curve .....	147
MW and Voltage Control .....	146
Generators .....	45
All Area Gen IC Curves .....	252
Auto Insert .....	91
Auxiliary Files.....	41
Case Information Display .....	197
Cost Case Information Display .....	199
Cost Curve Data Options .....	45
Cost Data Files .....	41
Cost Modeling.....	252
Cost Modeling.....	118
Display Objects.....	61
Edit Mode Dialog.....	114
<b>Fuel cost curve</b> .....	252
<b>Heat-rate curve</b> .....	252, 253
Incremental cost curve.....	252
Information Dialog.....	145
Input-output curve.....	252
Inserting and Placing Multiple Fields .....	55
OPF.....	321, 322
Outages.....	198, 199
Participation Factors Information .....	119
Reactive Capability Curve Files.....	41
Reactive Capability Curves .....	120

Records .....	197
Remotely Regulating.....	195
Representation on Onelines.....	61
Sorting.....	197
Total Available AGC Range.....	154
Transfer Distribution Factors .....	285
Getting Started .....	18
Graphic Formats.....	49

## H

Heat-Rate Curve .....	252
Help .....	17
HTML	
Editor.....	378, 379, 380
HTML Link Properties .....	380
Image Properties.....	380
Saving Case Information As.....	377
Table Format .....	377, 378

## I

Identify Cross Compound Units .....	45
<b>IEEE Common Format</b> .....	38
Impedance Correction Tables.....	152
<b>Incremental Cost Curve</b> .....	253
INF Files.....	41, 46
Information Dialogs	
Areas .....	57
Buses .....	142, 143, 144
DC Transmission Lines .....	128
Generators.....	145
Interface Fields .....	168
Interface Pie Charts .....	168
Loads .....	148
Multi-section Lines .....	126
Phase shifters.....	138
Super Areas.....	162, 163
Switched Shunts .....	153, 154
Transformer AVR.....	135
Transformer Fields.....	139
Transformers .....	149, 150, 151
Transmission Lines .....	149
Zones .....	57, 109, 155
INJ Files.....	41, 46
Injection Groups	
Contributions By Area .....	225
Creating.....	223
Deleting.....	223
Dialog.....	224, 225
Display.....	224
Format (*.inj).....	46
Overview .....	223
Saving/Loading .....	41
Input-Output Curve.....	147, 253
Insert Menu (Edit Mode) .....	54
Background Ellipses.....	75

Background Lines .....	74
Background Rectangles .....	75
Pictures .....	75, 76
Insert Palette .....	21
InterArea Flow Options Dialog .....	72
Interface Data Format (*.inf) .....	46
Interface Element Dialog .....	167, 168
Interface Fields	
Information Dialog .....	168
Representation on Onelines .....	72
Interface Pie Charts	
Information Dialog .....	168
Representation on Onelines .....	72
Interface Records .....	209
Interfaces	
Auto Insert .....	94
Automatically Defining .....	71
Auxiliary Files .....	41
Case Information Display .....	209
Contouring .....	281
Element Dialog .....	167
Fields on Onelines .....	72
Flowgates .....	73
Information Dialog (Edit Mode) .....	165
NERC .....	73
OPF Records .....	325
Pie Charts on Onelines .....	72
Representation on Onelines .....	70
Sensitivities .....	293
Transfer Distribution Factors .....	285
Introduction	
Edit Mode .....	18
PowerWorld Simulator .....	15
Run Mode .....	18, 19
Island Records .....	210
<b>J</b>	
Jacobian .....	50
Jacobian Display .....	220
Jpegs	
Exporting Onelines As .....	49
Exporting Strip Charts As .....	49
Options Dialog .....	50
<b>K</b>	
Key Fields .....	182
KV-Based Color Dialog .....	85
<b>L</b>	
Labels .....	155, 156
Limit Monitoring Settings Dialog .....	217
Limit Monitoring_Settings .....	216
Limit Violations	
Listing .....	214
Limiting Data Shown on Case Information Displays .....	176

Limits Options .....	237
Line Fields.....	55
Edit Mode Dialog.....	127
Representation on Onelines.....	64
Line Flow Interface Sensitivities.....	293
Line Flow Pie Charts .....	65
Line Outage Distribution Factors	
Line Outage Distribution Factors Dialog .....	289
Line Records.....	202
Line Shunt Information.....	125
Line/Fill	
Changing.....	104
Properties.....	106
Lines and Transformers	
Auto Insert.....	90
Case Information Display .....	202
Contouring.....	281
Edit Mode Dialog.....	123
Fields on Onelines .....	55, 64
Inserting and Placing Multiple Fields .....	55
Line Shunts.....	125
OPF Records.....	322
Outages.....	203, 204
Records.....	202
Run Mode Dialog.....	149
Sensitivities.....	293
Series Capacitors.....	125
Sorting.....	202
Transfer Distribution Factors.....	285
Links	
To Documents .....	74
To Other Onelines.....	73
List All Display Objects .....	88
List Display Palette.....	22
Load and Generation Chart .....	246
Load Schedules	
Load Schedule Dialog.....	164
Load Schedule Records.....	201
Loads	
Area Load Schedules.....	190
Auto Insert.....	92
Automatic Variation.....	122, 164, 201
Case Information Display .....	201
Case Load Multiplier .....	164, 201
Edit Mode Dialog.....	120
Information Dialog.....	148
Inserting and Placing Multiple Fields .....	55
Modeling.....	122, 201
Records.....	201
Representation on Onelines.....	62
Scaling.....	98, 99
Sorting.....	201
Zone Load Schedules .....	191
Local Menus	
Case Information Displays .....	173, 174
Online Diagrams .....	24

LODF	
Line Outage Distribution Factors Dialog .....	289
Log .....	19
Losses	
Calculation .....	285
Chart .....	246
Due to Marginal Transactions .....	285
Sensitivities .....	293, 294
Total for Case .....	186
<b>M</b>	
Make Reports .....	189
MATLAB	
Exporting Ybus and/or Jacobian .....	50
Merging cases .....	77
Message Log .....	19
Metafiles	
Exporting Onelines As .....	49
Mismatches .....	196
Model Criteria	
Model Conditions .....	180
Model Filters .....	180, 181
Modeling	
Loads .....	122
Relationship between Display Objects and Model Data .....	23
Transactions .....	243, 244
Transformers .....	133, 134
Modifying the appearance of objects .....	53
Mouse Usage .....	18
Movie Maker .....	253
Movie Maker Dialog .....	253, 254
Moving Display Objects .....	53
Multi-Section Line Records .....	204
Multi-Section Lines	
Case Information Display .....	204
Deleting .....	125
Information Dialog .....	125
Inserting .....	204
Sorting .....	204
Multi-Terminal DC Records	
Multi-Terminal DC Bus Information .....	131
Multi-Terminal DC Converter Information .....	132
Multi-Terminal DC Line Information .....	133
Multi-Terminal DC Record Display .....	207
Multi-Terminal DC Record Information .....	130
Mutual Impedance Record Dialog .....	280
Mutual Impedance Records .....	279
<b>MVA Base</b> .....	230
MVR Control	
Transformers .....	137
MW and Voltage Control .....	146
MW Distance	
Calculate .....	244, 245
Options .....	245

**N**

NERC  
     Flowgates .....73  
 New Case .....40  
 Newton Raphson Power Flow.....240

**O**

Obtain an Initial Solution.....298  
 Online Diagrams  
     Animation.....30  
     Bus View Displays .....247, 248  
     Closing .....41  
     Conditional Display of Objects.....34  
     Copying Objects .....101  
     Copying to Other Programs.....31  
     Creating New .....40  
     Cutting Objects .....101  
     Deleting Objects .....101  
     Exporting as Jpegs .....49  
     Finding Buses  
         Areas or Zones .....103  
     Links to other Onelines .....73  
     List All Objects .....88  
     Local Menu .....24, 25  
     Opening.....39  
     Options .....25  
     Other Graphic Formats .....49  
     Overview .....23  
     Pasting Objects.....101  
     Printing .....34  
     Saving .....41  
     Screen Coordinates .....31  
     Select by Criteria.....101  
     Showing Unlinked Display Objects .....87  
     Static Background.....74  
     Text Fields .....76  
     Zooming and Panning.....31  
 Online Display Options .....55  
 Online Display Options Dialog  
     Animated Flows Options .....28  
     Display Options.....26  
     Pie Chart Options.....27  
     Thumbnail View .....29  
 Online Links .....73  
 Online Popup Hints.....237  
 Online Templates .....51  
 Opening  
     Auxiliary Files.....41  
     Online Diagrams .....39, 73  
     Simulation Case.....39  
 OPF  
     Active Constraints .....312  
     Area Records.....323  
     Bus Marginal Price Details .....320  
     Bus Records .....320

Constraints .....	309
Controls .....	326
Equality Constraints .....	310
Example .....	327, 329, 330
Future Enhancements .....	314
Generators .....	321, 322
Inequality Constraints .....	311
Interface Records .....	325
Line and Transformer Records .....	322
Load LP OPF Data .....	332
LP Basic Variables .....	318
LP Basis Matrix .....	318
Marginal Costs .....	313
Objective Function .....	309
Options .....	315
Overview .....	308
Phase Shifter Controls .....	326
Primal LP .....	313, 314
Restore Previous Control Settings .....	327
Single Primal LP Outer Loop .....	314
Solution Results .....	318
Store LP OPF Data .....	332
Super Area Records .....	324
Unenforceable Constraints .....	312, 313
Options	
Case Information Display .....	229, 234, 235
Environment .....	229, 232
Limits .....	229
Online Diagrams .....	25
OPF .....	315
OPF Bus Marginal Price Details .....	320
OPF Constraint Options .....	315
OPF General Options .....	316
OPF LP Basic Variables .....	318
OPF LP Basis Matrix .....	318
OPF Solution Results .....	318
Power Flow Solution .....	229
Simulation .....	229, 235, 236, 237
Solution/Environment .....	229
Template .....	229, 235
Options Menu	
Area Average Cost Chart .....	247
Area Load and Generation Chart .....	246
Area Losses Chart .....	246
Area MW Transactions Chart .....	247
Options Menu (Edit Mode)	
Append Case .....	77
Create Equivalent .....	96, 97
Default Drawing Values .....	104
List All Display Objects .....	88
Show Unlinked Display Objects .....	87
Options Menu: .....	55
Options/Info Palette .....	22
Other Applications	
Exchanging Data With .....	181
Outages	

Generators.....	198, 199
Lines and Transformers .....	203
Overview.....	15
Edit Mode .....	53
Online diagrams.....	23
Project File.....	373
Owner	
Bus Records .....	220
Dialog.....	219
Generator Display .....	220
Information Display .....	219
Line Display.....	220
Load Display.....	220
Update Owners Dialog.....	220
Ownership.....	219
<b>P</b>	
Page Up/Down Keys	
Zooming and Panning.....	31
Palette.....	88
Edit.....	21
File .....	21
Format.....	21
Insert .....	21
List Display .....	22
Options/Info .....	22
Overview .....	88
Pan/Zoom.....	22
Program.....	20
Run Mode.....	22
Using .....	89
Zoom .....	22
Palettes.....	20
Pan/Zoom Palette.....	22
Panning.....	31, 32
To a Particular Bus	
Area or Zone .....	103
<b>Participation Factor Control</b> .....	241, 242
Participation Points	
Add.....	227, 228
Overview .....	226
Records Display.....	226, 227
Paste Command.....	108
Pasting Objects .....	108
Pause.....	239
<b>Pausing</b> .....	239
Phase Shifters.....	138
Pictures	
Inserting on Online.....	75
Pie Chart Options .....	27
Pie Charts	
Interfaces.....	72, 168
Transmission Lines/Transformers .....	65
Power Flow	
Algorithms.....	240
Described .....	240

Jacobian .....	50
Newton Raphson .....	240
Power Flow List .....	187
Power Flow Solution Options.....	229
<b>Area Interchange</b> .....	232
Automatic Controls.....	231, 232
General.....	230
Storage.....	232
Power Transfer Distribution Factors (PTDFs).....	285
Auto Insert Directions.....	292
Contouring Options .....	281
Directions Dialog.....	291
Directions Display .....	292
PTDF Calculation Dialog.....	285
PowerWorld Corporation .....	17
PowerWorld Project File	
Associating with Simulator .....	374
Create Project Dialog.....	375
Creating.....	375
Initialization.....	373
Opening.....	376
Overview .....	373
Print Options Dialog.....	34
Printer Setup .....	39
Printing	
Online Diagrams .....	34
Options Dialog .....	34
Printer Setup.....	39
Strip Charts.....	34
Program Palette .....	20
Project File	
Associating with Simulator .....	374
Create Project Dialog.....	375
Creating.....	375
Initialization.....	373, 374
Opening.....	376
Overview .....	373
PTDF .....	285
Auto Insert Directions.....	292
Calculation Dialog .....	285
Contouring.....	281
Directions Dialog.....	291
Directions Display .....	292
PTI Raw Data Files.....	37
Publish to Web Dialog .....	381
Publishing to web .....	19, 378
PVQV	
Consider What to Do.....	298
Define Regions to Monitor.....	300
Defining Contingencies .....	300
Defining Injection Groups.....	298
Defining What to Monitor.....	302
How Should the System Vary.....	301
Load State Pause Restore Orginal .....	306
Open the PV Study Form .....	300
Options to Control the Transfer Increase .....	301

Other Options .....	304
Performing the Analysis .....	305
PowerWorld Simulator VAST Overview .....	297
Q-V Curve Buses .....	306
Q-V Curve Control/Results .....	308
Q-V Curve Options.....	307
Q-V Curves.....	306
Refine Model.....	308
Save Options for Future Use .....	305
Setting Output Options.....	304
The Output File .....	306
PWB Files .....	38

**Q**

Quick Power Flow List .....	188
Bus View Display .....	247, 248

**R**

Ranges of Numbers.....	181
Reactive Capability Curves.....	120
Auxiliary Files.....	41
Reactive Power Control	
Transformers .....	137
Recently Opened Cases.....	40
Rectangle Selector .....	102
Relationship Between Display Objects and Power System Model .....	23
Remotely Regulated Bus Records .....	194
Renumbering Buses .....	77
Reports	
Area Reports.....	189
Criteria Dialog.....	189
Zone Reports.....	189
Reset .....	239, 240
Resetting.....	238
Resizing Display Objects .....	53
Run Mode	
Introduction.....	18
Run Mode Palette.....	22

**S**

Save View Dialog .....	32
Saving	
Auxiliary Files.....	41
Case Information Data .....	182
Cases .....	40
Onelines .....	41
Saving Favorite Views .....	32
Scaling .....	98
Scheduled Transactions	
Modeling.....	243
Setting Up.....	213
Schedules	
Auxiliary Files.....	41
Disabling.....	201
Load Variation.....	41, 164, 165, 201, 202

SCP Files .....	343
Screen Coordinates .....	31
Script Files .....	343
Data Argument List .....	358, 359
Data ATCScenario .....	370
Data ColorMap .....	362
Data Contingency .....	362
Data CTG Options .....	365
Data DataGrid .....	370
Data Filter .....	365
Data Gen .....	366
Data InjectionGroup .....	367
Data Interface .....	368
Data Key Fields .....	360
Data List .....	360
Data ModelCondition .....	368
Data ModelFilter .....	369
Data MultiSectionLine .....	367
Data Owner .....	369
Data Section .....	358
Data SubData Sections .....	361
Script ATC Submode Actions .....	354
Script Contingency Submode Actions .....	353
Script Edit Mode Actions .....	348
Script Fault Submode Actions .....	355
Script General Actions .....	345
Script PowerFlow Submode Actions .....	353
Script PV Submode Actions .....	356
Script QV Submode Actions .....	357
Script Run Mode Actions .....	350
Script Section .....	344
Select By Criteria .....	102
Selecting Objects .....	53
All Objects in Region .....	102
By Criteria .....	102
By Object Type .....	102
Sensitivities	
Flows and Voltages .....	293
Losses .....	293, 294
Transmission Loading Relief Sensitivities .....	295
Sequence Data Format .....	47
Series Capacitor Information .....	125
Series Capacitors	
Information Dialog .....	128
Representation on Onelines .....	68
Set Custom Detail Dialog .....	30
Set Generator Participation Factors .....	242
SHD Files .....	41
Shortest Path Between Buses .....	113
Shunts	
Information Dialog .....	153
Inserting and Placing Multiple Fields .....	55
Representation on Onelines .....	69
Simulation Options .....	235
Simulation/Environment Options	
Environment Options .....	232

Simulator	
Getting Started.....	18
Introduction.....	15
Overview .....	15
<b>Single Solution</b> .....	240
Slack Bus(es)	
Listed.....	186
Listed by Island.....	210
Snap-to-Grid.....	103
Solution/Environment Options.....	229
Case Information Display Options .....	234
Limits Options.....	237
Popup Hints Options .....	237
Power Flow Solution Options .....	229
Simulation Options.....	235
Template Options.....	235
Solving the Power Flow .....	240
Mismatches .....	196
Sorting	
Area Records.....	190, 191
Bus Records .....	194
Case Information Displays .....	176
DC Line Records.....	207
Generator Cost Records .....	199
Generator Records .....	197
Interfaces.....	209
Line/Transformer Records.....	202
Load Records .....	201
Mismatches .....	196
Multi-Section Line Records .....	204
MW Transaction Records.....	211
Remotely Regulated Bus Records.....	194
Super Area Records.....	192
Switched Shunt Records.....	208
Transformer Records .....	205
Zone Records .....	192
Splitting Buses .....	79, 80, 81
Equipment Mover.....	81
Potential Misplacements Dialog.....	82
Splitting Buses.....	79
<b>Stack Level</b> .....	107
<b>Start</b> .....	240
Starting .....	239
Strip Chart Displays	
Area Average Cost.....	247
Area Load and Generation .....	246
Area Losses.....	246
Area MW Transactions.....	247
Exporting as Jpegs .....	49
<b>Study System</b> .....	96
Substations	
Automatically Inserting.....	94
Display Objects.....	61
Edit Mode Information.....	113
Records Display.....	196
Run Mode Information.....	144

Substation View Display.....	248, 249
Super Area	
Field Information .....	163
Information Dialog.....	162, 163
Inserting Fields .....	59
OPF Records.....	324
Records.....	192, 193
Sorting.....	192
Switched Shunts	
Auto Insert .....	93
Case Information Display .....	208
Edit Mode Dialog.....	140
Information Dialog.....	153
Representation on Onelines.....	69
<b>T</b>	
Tapping Transmission Lines.....	82
Automatic Line Tap Dialog .....	82, 83
Technical Support .....	17
Template Options.....	235
Templates	
Case Templates.....	51
Online Templates.....	50, 51
Overview .....	50
Text Displays	
Power Flow List .....	187
Quick Power Flow List.....	188
Text on Onelines .....	76
Three Winding Transformer Display.....	206
Three Winding Transformer Information.....	139
Thumbnail View.....	29
TLR Sensitivities Dialog.....	295
Toolbar.....	33, 34
Toolbars.....	20
Edit Palette .....	21
File Palette.....	21
Format Palette .....	21
Insert Palette.....	21
Options/Info Palette .....	22
Pan/Zoom Palette .....	22
Program Palette.....	20
Run Mode Palette .....	22
Zoom Palette .....	22
Transaction Records .....	211
Transactions	
Base/Scheduled Interchange Defined .....	243
Chart .....	247
Distribution Factor Calculation.....	285
Information Dialog.....	213
Modeling of .....	243
Records .....	211
Setting Scheduled Transactions.....	213
Sorting.....	211
Wheeling .....	214
Transfer Distribution Factors.....	285
Transformer Fields	

Information Dialog.....	139
Representation on Onelines.....	67
Transformer Impedance Correction Tables	
Case Information Display .....	135
Modifying .....	152
Using with a Phase Shifting Transformer.....	138
Using with an LTC Transformer.....	135
Transformer Records.....	205
Transformers	
Adding an Impedance Correction Table.....	135
AVR Information Dialog.....	135
Case Information Display .....	205
Fields on Onelines .....	67
Impedance Correction.....	135
Impedance Correction Table Dialog .....	152
Information Dialog.....	149
Modeling.....	133
Mvar Control Dialog .....	137
MW and MVAR Control.....	133
Phase Shifter Information Dialog .....	138
Records .....	205
Representation on Onelines.....	66
Sorting.....	205
Types of Control .....	133
Voltage Control .....	134
Transmission Lines	
Fields on Onelines .....	64
Information Dialog.....	149
Line Shunts.....	125
Representation on Onelines.....	63
Transmission Loading Relief Sensitivities .....	295
Tutorial	
Adding a Background.....	395
Adding a New Bus .....	385
Adding a New Generator.....	386
Adding a New Load.....	387
Adding a New Transmission Line .....	387
Adding an Existing Bus to a Oneline.....	395
Adding New Area Objects .....	391
Adding New Text Fields .....	389
Area Records.....	396
Automatically Inserting Lines.....	395
Building Onelines .....	394
Case Information Displays .....	394
Creating a New Case .....	385
Creating a Oneline for an Existing Model.....	393
Customizing the Case .....	392
Information Dialogs .....	396
Limit Violations.....	396
Loading an Existing Power Flow Model .....	393
Oneline Local Menu .....	396
Other Case Information Displays.....	397
Panning and Zooming .....	395
Saving a Case .....	386
Simulating a Case.....	396
Solving the Case.....	390, 394

**U**

Unlinked Display Objects.....	87
Update Owners Dialog.....	220
Using the Toolbars .....	20
Edit Palette .....	21
File Palette.....	21
Format Palette .....	21
Insert Palette.....	21
Options/Info Palette .....	22
Pan/Zoom Palette .....	22
Program Palette.....	20
Run Mode Palette .....	22
Zoom Palette .....	22

**V**

Validate Case.....	38
Var Limits.....	120
Version 8.0 New Features .....	16
Views	
Saving Favorite Views.....	32
Violations	
Listing.....	215
Voltage	
Contouring.....	281

**W**

Web Publishing .....	377
Adding Attachments.....	382
Define New Attachment Dialog.....	382
HTML Editor .....	378
HTML Image Properties .....	380
HTML Link Properties .....	380
Overview .....	377
Publish to Web Dialog.....	381
Saving Case Info Displays as HTML .....	377
Saving Images as Jpegs .....	49
Sending documents to your web server.....	381
Table Format .....	377
Web Publishing Status.....	383
What is New in Version 8.0.....	16
Wheeling	
Wheeling Areas Dialog.....	214
<b>Window</b>	
<b>Cascade</b> .....	33
<b>Refresh Displays</b> .....	33
<b>Show Full</b> .....	33
<b>Tile</b> .....	33
<b>Toolbars</b> .....	33
World-wide web pages .....	19

**Y**

Ybus.....	50
Ybus Display .....	221

**Z**

Zone Fields .....	58
Zone Records.....	191
Zones	
Area/Zone Display Options .....	56
Area/Zone Filters .....	176
Case Information Display .....	191
Changing.....	109
Fields on Onelines .....	57
Finding on the Onelines .....	103
Information Dialog.....	109, 154
Load Schedules .....	191
Load Variation Records.....	201
Marginal Losses for a Transaction.....	285
Records .....	192
Representation on onelines.....	56
Representing on Onelines.....	56
Sorting.....	191
Transfer Distribution Factors .....	285
Zoom Levels.....	105
Conditional Display .....	34, 104
Zoom Palette.....	22
Zoom/Pan/Find Dialog.....	103
Zooming and Panning .....	31
Options .....	103