POWERWORLD Simulator

Version 6.0

Interactive power system simulation, analysis and visualization

User's Guide

POWERWORLD Corporation

September 1999

POWERWORLD Simulator
License Agreement

Please Read Carefully Before Use

POWERWORLD CORPORATION ("LICENSOR") IS WILLING TO LICENSE THE SOFTWARE WHICH ACCOMPANIES THIS LICENSE TO YOU ONLY IF YOU ACCEPT ALL OF THE TERMS IN THIS LICENSE AGREEMENT. PLEASE READ THE TERMS CAREFULLY. IF YOU DO NOT AGREE TO THESE TERMS, LICENSOR WILL NOT LICENSE THIS SOFTWARE TO YOU, AND IN THAT CASE YOU SHOULD RETURN THIS PRODUCT PROMPTLY TO THE PLACE OF PURCHASE FOR A FULL REFUND.

Ownership of the Software

1. The software which accompanies this license (the "Software") and the accompanying written materials are owned by POWERWORLD CORPORATION (POWERWORLD) [or its suppliers] and are protected by United States copyright laws, by laws of other nations, and by international treaties.

Grant Of License

2. Licensor grants to you the right to use one copy of the Software on a single computer. You may load one copy into permanent memory of a single computer. You may install the Software on a single network server, provided there is no possibility of the Software being used by more than one user simultaneously.

3. The primary user of the computer on which the Software is installed may make a second copy for his or her exclusive use on either a portable or a home computer.

Restrictions on Use and Transfer

4. You may not copy the Software, except that (1) you may make one copy of the Software solely for backup or archival purposes, and (2) you may transfer the Software to hard disks as specified in above under Grant Of License.

5. You may not reverse engineer, decompile, or disassemble the Software.

Limited Warranty

6. POWERWORLD warrants that the Software will perform substantially in accordance with the accompanying written materials for a period of 60 days from the date of your receipt of the Software. Any implied warranties on the Software are limited to 60 days. Some states do not allow limitations on duration of an implied warranty, so the above limitation may not apply to you.

7. POWERWORLD DISCLAIMS ALL OTHER WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT, WITH RESPECT TO THE SOFTWARE AND THE ACCOMPANYING WRITTEN MATERIALS. This limited warranty gives you specific legal rights. You may have others, which vary from state to state.

8. POWERWORLD'S ENTIRE LIABILITY AND YOUR EXCLUSIVE REMEDY SHALL BE REPLACEMENT OF THE SOFTWARE THAT DOES NOT MEET LICENSOR'S LIMITED WARRANTY AND WHICH IS RETURNED TO LICENSOR. Any replacement Software will be warranted for the remainder of the original warranty period or 30 days, whichever is longer. These remedies are not available outside the United States of America.

9. This Limited Warranty is void if failure of the Software has resulted from modification, accident, abuse, or misapplication.

10. IN NO EVENT WILL LICENSOR BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY LOSS OF PROFITS, LOST SAVINGS, OR OTHER INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF YOUR USE OR INABILITY TO USE THE SOFTWARE. Because some states do not allow the exclusion or limitation of liability for consequential or incidental damages, the above limitation may not apply to you.

11. This Agreement is governed by the laws of the State of Illinois.

12. If you have any questions concerning this Agreement or wish to contact POWERWORLD for any reason, please email info@powerworld.com or call (217) 265-0286.

13. U.S. Government Restricted Rights. The Software and documentation are provided with Restricted Rights. Use, duplication, or disclosure by the Government is subject to restrictions set forth in subparagraph (c)(1) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c)(1)(ii) and (2) of Commercial Computer Software - Restricted Rights at 48 CFR 52.227-19, as applicable. Supplier is POWERWORLD CORPORATION, 2004 South Wright St., Urbana, IL 61801.

Table of Contents

CHAPTER 5 : EDITTING, FORMATTING, AND CUSTOMIZING ONELINE DIAGRAMS 69

CHAPTER 6 : PROPERTIES OF ONELINE DISPLAY OBJECTS   79

CHAPTER 7 : SOLVING AND SIMULATING A CASE   111

CHAPTER 8 : TEXT-BASED INFORMATION DISPLAYS   141

CHAPTER 9 : INJECTION GROUPS   181

CHAPTER 10 : EDIT MODE TOOLS AND OPTIONS   187

CHAPTER 11 : RUN MODE TOOLS AND OPTIONS   201

Chapter 1

Getting Started with PowerWorld

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

The following material is included:

- Introduction to PowerWorld Simulator

- About this Manual

- Windows Basics

- What's New in Version 6.0

- 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 6.0 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, 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.

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.

Windows Basics

Simulator 6.0 runs under Windows 95/98 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 effect 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.

What's New in Version 6.0

Simulator Version 6.0 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:

Ownership Data

Simulator now allows you to store ownership information for buses, loads, generators, and transmission lines. Generators and transmission lines can have up to four owners, each with their own specified percentage of ownership. Loads and buses can have one owner.

Saving Views

The ability to store various views for a oneline diagram has been added to Simulator. The saved views option allows moving around to specific locations on the diagram even quicker than before by saving a list of the x and y coordinates, as well as the zoom level.

Automatically Insert Loads, Generators, and Interfaces

Adding on to Simulators automatic line inserting feature, Simulator 6.0 adds the ability to automatically insert loads, generators, and interfaces. These features can greatly decrease the amount of time it takes the user to construct a oneline diagram for an existing case.

Outage Information

Simulator 6.0 makes it extremely easy to view the generators, lines, and transformers that are out of service. The outage information is displayed in the form of a table and can be viewed any time in Edit Mode or Run Mode.

Super Areas

A super area is a collection of areas that function as one area. Simulator 5.0 allowed the user to create a super area for modeling in the Power Flow Solution. Simulator 6.0 now takes this feature one step further and allows super areas to be displayed on the oneline diagram.

Bus Palette

The bus palette is designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. It lists the buses that you have already added to the oneline and the set of all buses that have not yet been added to the drawing. With the new bus palette, buses can be added to the drawing with very little effort.

Contingency Analysis

PowerWorld Simulator 6.0 has added several new features to our contingency analysis in order to make it much more user-friendly. The ability to add a combination of up to three transformers, three lines, and three generators has been added to the automatic insert contingency dialog. The contingency reports have also been enhanced with the ability to view the report immediately after being created and creating three delimited text files for convenient transfer into a database or a spreadsheet.

Injection Groups

An injection group is a collection of loads and generators, and each load or generator can belong to more than one injection group. Moreover, a single injection group may contain generators and loads from several different areas and zones. 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.

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 (page 249). Or if you're interested in trial-and-error learning, you may just wish to open one of the sample cases and start learning.

Using the Toolbars

Simulator 6.0 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. The toolbar palettes are identified here.

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 (page 118) 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 (page 118) 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 (Edit Mode only)

The Edit Palette 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 (Edit Mode only)

The Insert Palette 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 branch of the main menu.

Format Palette (Edit Mode only)

The Format Palette 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.

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 (page 26) 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 display 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 and difference flows.

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.

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

- Create a completely new case; see New Case (page 37) for details.

- Create a new oneline diagram; see New Oneline (page 38) for details.

- Add new components graphically to an existing case; see Insert Menu (page 49) for details.

- Modify the appearance of the oneline objects; see Format Menu (page 72) for details.

- View and modify a case using non-graphical lists displays; see Case Information Displays (page 146) for details.

- Equivalence a case; see Equivalencing (page 123) for details.

- Append a subsystem to an existing case; see Appending a Case (page 188) for details.

For a more details on the Edit Mode please see Edit Mode Overview (page 48).

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.

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 (page 23) for details.

- The Case Information Displays, which allow you to view the entire power system case using list displays. See Case Information Displays (page 146) for details.

- Dialogs to change the simulation options and the Power Flow Solution. See Simulation Options (page 111) for details.

- Various strip-charts for plotting the time-variation in system values. See Load and Generation Chart (page 125) for an example.

- Scaling to allow easy variation in the load, shunts, and generation at any number of buses. See Scaling (page 198) for details.

- Contouring, which shows a color contour representing the variation in any power system parameter across a system. See Contouring (page 127) for details.

- Transfer distribution factor calculations. See Power Transfer Distribution Factors (page 132) 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 (page 231) 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 (page 233) for details.

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

Getting 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* 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

Chapter 2

Using Oneline Diagrams

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

The following material is included:

- Oneline Diagram Overview

- Oneline Screen Coordinates

- Oneline Local-Menu

- Save View Level Dialog

- Oneline Zooming and Panning

- Oneline Animation

- Print Options Dialog/Printer Setup

- Oneline Conditional Display of Objects

- Copying Onelines to Other Programs

- Window Menu Command

- Oneline Customization Options

- Relationship Between Display Objects and the Power System Model

Oneline Diagram Overview

The purpose of the oneline diagram is to show information about the power system graphically. Such displays are called oneline diagrams (onelines) because the actual three-phase power system components are represented using a single line. Simulator 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 oneline diagram. For most objects, this is accomplished by positioning the cursor on the object and right-clicking.

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

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 (page 26).) Negative screen coordinates are allowed. Usually you will not have to be concerned about an object's location in screen coordinates.

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 (page 28) 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 (page 29) 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 (page 39)) or a oneline template (see Oneline Templates (page 39)) 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.

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

Y-Coordinate

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

Zoom Level

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

Save

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**.

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 (page 28) 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 Oneline** 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.

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 Oneline 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.

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 it's local menu, and then select **Print Window**. In both cases, Simulator will open the Print Options Dialog.

The Print Options Dialog has two tabs. The first tab is labeled **Page Layout**, and it controls 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 online 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 online 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

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.

Printing Oneline Diagrams

To print a online diagram, select **File, Print Oneline** from the main menu, click the **Print Window** button on the File Palette, or right-click on the online background and select **Print Window** from the popup menu. This opens the Print Options Dialog, which you can use to configure the print job, including its size, orientation, border, and title bar.

Oneline Conditional Display of Objects

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

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

This option is available from **Format, Display/Zoom Levels**.

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.

Windows 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 oneline diagrams such that the total window area is divided equally among all of them and each is completely visible.

Cascade

Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

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 oneline diagram so that all of the objects contained on the oneline 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.

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*, and *Animated Flows Options*.

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:

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

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

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.

Use Absolute Values for MW Line and Interface Flows

If checked, this option will cause all MW flow text fields for lines and 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 line or interface and negative near the sink end of the line or 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 Max. 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 highest 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 Color

The standard, pre-warning fill color 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 (page 150) 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 (page 150) for more information.

Only in Monitored Areas

If this option is checked, then only those pie charts that correspond to branches in areas for which limit violations are supposed to be reported 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.

<div align="center">

Animated Flows Options

</div>

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 Symbol

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

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 this field or click the **Change Fill Color** button to change the fill color.

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.

Max Flow MVA

This field is used to scale how far the animated flows should move between each screen refresh. This field specifies the MVA flow corresponding to the maximum change in animated symbol position. The maximum change in the location of a symbol from one time step to the next is 0.35 times the distance between the symbols. This limit is necessary because larger changes could cause the line flow to appear to be flowing in the opposite direction. In essence, this field controls the speed of the animated flows. All MVA flows that exceed the value specified in this field will be represented by an animated flow of equal speed.

Flow Visualization

Animated flows on onelines may depict either actual power flows or power transfer distribution factors, depending on what you choose here.

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.

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 provided access to many of the same options housed by the File Menu.

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 (page 37) 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 (page 47) 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.

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. CAUTION: when opening a PTI Raw Data Format file, you must know whether it is version 23 or version 24/25/26; typically all versions use the same *.raw extension. When using this format, the generator participation factors are automatically set to be proportional to the MW rating for the generator.

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.

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.

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.

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 or 25 raw data, and the IEEE common format. Please see Case Formats (page 37) 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.

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.

Saving a Oneline

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

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

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

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.

- **Oneline Display Templates**, which allow you to apply the options used for a particular oneline diagram to a different oneline diagram.

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

For more information, see Case Templates (page 39) and see Oneline Templates (page 39).

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.

Oneline Templates

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

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.

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 he 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 (page 231) 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.

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 (page 196)) 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*, and load schedules with the extension *.shd*. These additional file types are for compatibility with auxiliary files generated in previous versions of PowerWorld Simulator.

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

auxfiletype	an identifier for the type of data stored within the auxiliary file,
identifiers:	AREA/ZONE LOAD SCHEDULES
	AREA AGC, AUTO TAPS, AUTO SHUNTS
	GENERATOR REACTIVE CAPABILITY CURVES
	GENERATOR COST DATA
	PARTICIPATION GROUPS
	INTERFACE RECORDS
auxfiledata	the data for the file type, in the same formats as the format descriptions for the individual file types below.

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.

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 Schedules (*.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 1.500   9:00 2.5000 10:00 1.5000

-1

Automatic Area Controls (*.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

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

Generator Capability Curves (*.gcc)

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, P_1, $Q_{1,max}$, $Q_{1,min}$, P_2, $Q_{2,max}$, $Q_{2,min}$ ... P_n, $Q_{n,max}$, $Q_{n,min}$**

where

num/name	the generator's bus number or the bus' name in single quotes,
ID	the generator's single character id,
P_i	a generator MW output value,
$Q_{i,max}$ and $Q_{i,min}$	the associated maximum and minimum reactive power limits.

Up to 10 different $P/Q_{max}/Q_{min}$ values may be specified.

Generator Cost (*.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.

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 'C' or is not specified, the record describes a cubic cost model and thus concludes with the following fields:

a_i, b_i, c_i, d_i, **FuelCost**

where

a_i, b_i, c_i, d_i input-output curve coefficients; see Generator Cost Information (page 93) for details,

fuelcost   fuel cost, expressed in price per MBtu

If, on the other hand, CostModel is 'P', the record describes a piecewise linear cost model. The remainder of the record specifies pairs of generator output and corresponding generator cost:

FixedCost, MW_1, $Cost_1$, MW_2, $Cost_2$, ... , MW_n, $Cost_n$, MWneg

where

FixedCost   Operation cost independent of the generator's MW output; expressed in $/MWHr,

MW_i, $Cost_i$ Output/cost data point pairs that define the piecewise linear cost model for the generator,

MWneg   An arbitrary negative value used to terminate the record.

Injection Groups (*.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

groupname   the name of the injection group, in single quotes,

devtype   either GEN or LOAD,

busnum   bus number of the device,

id   the device's single character ID,

participation   relative amount each device will contribute during a transfer,

pointtype   either FIXED or DYNAMIC.

Multiple GROUP sections can be put in the file, each marked with END to signify the end of that group's data.

Interface Data

The interface data files store the interface data. The files have the following format:

INTERFACE intname lima limb limc

Elemtype from to id checktoend fromto

.

.

.

where

intname	the interface name, in single quotes,
lima, limb, limc	the three limits for the interface, in MW,
elemtype	one of three valid element types: AREA, LINE, or ZONE
from	for AREA and ZONE, the first area or zone number; for LINE, the from bus number,
to	for AREA and ZONE, the second area or zone number; for LINE, the to bus number,
id	only included for LINE, the two character circuit identifier; otherwise left blank,
checktoend	only included for LINE, if this field equals 1, the flow is checked at the to bus, otherwise the flow is checked at the from bus; otherwise left blank,
fromto	only included for LINE, if this field equals 1, the positive flow is assumed to be from the from bus to the to bus, otherwise the flow is assumed to be in the opposite direction; otherwise left blank.

Multiple interface records can be listed in the file, and multiple interface elements can be listed in each interface record.

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.

<p align="center">Ybus in MATLAB Format</p>

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.

Close Oneline

To close a oneline diagram, go to **File, Close Oneline**. This will prompt you to save the oneline if changes have been made since the last save. Note that this does not close the current case, just the oneline. Case simulations can be run with the oneline closed.

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

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.

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 (page 33) 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 display's the objects local menu, from which you can elect to see the object's dialog box.

Selecting Several Objects to Modify Their Appearance

Hold down the Shift key while clicking objects on the screen to select several objects at once. You may then change the objects' attributes, such as 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.

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 (page 29) for more information.

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 (page 146) 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.

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.

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 following choices are available:

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 (page 160) 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 Options. 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.

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. 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 (page 192).

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 (page 77) 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, Oneline Display, Default Font** to change the font for all fields on a display.

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.

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.

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.

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.

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 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, Oneline 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 (page 77) for more details.

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 (page 101) 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 transformers. 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° and 30°), and the step size in degrees (typical values are between 1° and 2°).

Select **OK** to save the values and return to the Transformer Options Dialog; otherwise select **Cancel**.

If you would like the transformer to be initially modeled as being on automatic control at the start of the case, select the *Automatic Control Active* checkbox.

If you do not want to add the transformer to the case, select Cancel.

To modify the parameters for an existing transformer, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the Line/Transformer Dialog. Use the Line/Transformer Dialog to adjust many of the transformer's electrical properties. Select **Format, Display Appearance** to change the transformer's color and/or line thickness.

See Transformer Modeling (page 101) 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, Oneline Display, Default Font** option to change the font for all fields on the display.

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.

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 Display Filters Set

If this option is checked, the set of potential areas or zones for inserting interfaces is limited to those for which the display 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 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 (page 190) 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 (page 108) 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.

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.

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 Oneline 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.

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.

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.

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 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.

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 (page 36) 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.

Chapter 5

Editing, Formatting, and Customizing Oneline Diagrams

The Format Menu is used in the Edit Mode to customize the appearance of the oneline 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

- Copying Simulator Data to and from Other Applications

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 oneline diagram.

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 (page 33) 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 (page 33) 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 (page 33) 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 (page 33) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the Default Drawing Options Dialog.

Unlike Cut, **Delete** does not copy the selection into the paste buffer.

Select by Criteria Dialog

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.   Finally, select the **Select All Except What Meets the Above Criteria** if that is what you wish to do.

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 <u>AND</u> 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.

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.

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 **Online 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.

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.

Display Appearance

The Display Appearance menu option provides a more general formatting tool. It simply invokes the Format Selection Dialog, from which the font, line/fill, conditional zoom, and object display attributes can be changed.

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.

Online Display

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

Default Font

Use this option to specify the oneline diagram's default font. Changing the default font will change the font used for all text objects on the display whose font has not already been changed using the Format Selection Dialog.

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.

Zoom/Pan

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, Oneline Display, Default Font**.

By clicking on **Set All to Default**, all the selected objects will be set to the default font name, font size, and font effects.

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.

Make Default for New Line Objects

Changes the default line settings so that any additional lines drawn on the display will automatically exhibit the selected features.

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 oneline.

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.

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.

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 *Buses* to find a bus object, *Areas* to find an area object, or *Zones* to find a zone object.

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.

Oneline Zoom Level and Screen Center

The Oneline Zoom Level and Screen Center section of the 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 (page 26) 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. Most of these values are updated dynamically when you create a new object and initially select a non-default size value.

See Setting Default Drawing Options (page 194) for more information.

Inserting and Placing Multiple Display Fields

Simulator 6.0 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.

The dialogs 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 top portion of the dialog 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 #*. The next section of the dialog is used to assign field types to these positions. For example, suppose you want to add a voltage field in position 1 for a bus display object. From the *field type* dropdown box situated next to the label 'Pos1', select 'Voltage'. Then, specify the total number of digits and the number of digits to the right of the decimal using the adjacent spin controls. Bus field position #1 will then specify the bus voltage to the precision you have specified. You may then assign other field types to other field positions around the bus. Adding fields to other display object types proceeds in an identical manner. Of course, the types of fields you can add varies with each object type.

Four buttons occupy the bottom of the dialogs used to insert and place multiply field objects. To employ the just defined field placement for all devices of the same type that will be added, click the *Set as Default for New Objects* button. This changes the default fields that will be added whenever a new object of the same type is dropped on the oneline. To restore the previous set of default fields, click the *Load Default for New Objects* button. The previous default fields set will be applied to the current display object and to all subsequently added display objects of the same type. Click *OK* to implement your choices for field additions and placement, or click *Cancel* to discard the changes.

It should be noticed from this discussion that all objects have an associated set of default fields that will be added to the oneline with them. You may redefine the default fields set as just discussed by pressing the *Set as Default for New Objects* button. Alternatively, select **Options/Tools, Default Drawing Values** from the main menu and press the *Set Default Fields Adding* button on the resulting Default Drawing Options Dialog. This brings up a five-page dialog, each page of which can be used to define field additions and placement for a particular display object type (bus, generator, line/transformer, load, and shunt).

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.

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 must match the case information display you are attempting to paste into, and the data headings tell Simulator which columns of data you are pasting. 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 date from the spreadsheet. Once you have copied all the information from the spreadsheet you wish to transfer, switch to Simulator, open the case information display in which to paste the data, and select **Paste** from its local menu.

Note: you can only paste values into the case information displays if the values are enterable on the display (shown blue by default). Also, be careful about pasting redundant data. For example, in the Bus Records display both voltage in *per unit* and voltage in *kV* are enterable, but they specify the same information. Make sure you only copy ONE of these columns into Simulator. Otherwise you may not get what you expect. Simulator will paste the value in twice, and whatever value was pasted 2nd will show up.

Chapter 6

Oneline Display Object Information

This chapter provides a description of the properties of each of the Oneline 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.

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.

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, which are described here:

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 (page 121) 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 (page 121).

Area Control Error

Current area control error (ACE) for the area in MW. Note that, when the constant frequency model is used, ACE = area generation - area load - area losses - 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.

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.

<div align="center">

Tie Lines

</div>

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.

<div align="center">

Simulation Summary

</div>

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 (page 121).

Area Control Error

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

Inadvertent Interchange

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 (MW)

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 (page 121) for more information on the difference between Study Transactions and Scheduled Transactions.

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 (page 162).

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 (page 160) for more detail.

AGC Status

Displays the AGC status of the area.

Select a Field

Choose from 40 different choices.

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.

Display Size

Specifies the length of the bus if its orientation is horizontal and height of the bus if orientation its orientation is vertical.

Pixel Thickness

Thickness of the bus in pixels.

Orientation

Either horizontal or vertical.

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.**

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.

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 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 display 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 display 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:

Bus Name	Name of the bus
Bus Number	Bus number (from 1 to 99,999)
Bus Voltage	Bus voltage magnitude
Bus Angle	Bus voltage angle in degrees
Load MW	Total MW load at the bus
Load Mvar	Total Mvar load at the bus
Switched Shunt Mvar	Total Mvar capacitance at the bus
Actual Gen MW	Total MW generation at the bus
Actual Gen Mvar	Total Mvar generation at the bus
Setpoint Gen MW	Only used with uniform frequency option
Gen AGC Status	AGC status of generator; status can be toggled in Simulator
Gen AVR Status	AVR status of generator; status can be toggled in Simulator
Mech Gen MW	Only used with uniform frequency option
MW Marginal Cost	Bus MW marginal cost in $/MWh; available only with OPF
Mvar Marginal Cost	Bus Mvar marginal cost in $/Mvarh; available only with OPF
MW Loss Sensitivity	Increase in losses for 1 MW of generation over the nominal
Gen Setpoint Voltage	Voltage of generator on bus
Select a Field	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.

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 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 this 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 this button.

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.

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.

Display Information

Display Size

The size of the generator.

Pixel Thickness

Thickness of the display object in pixels.

X and Y Location

X/Y location of the upper left corner of the display object.

Orientation

Specifies the direction in which to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See Anchored Objects (page 49) for details.

Input-Output Curve, Fuel-Cost, and Marginal Cost Grouping

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 (page 93) 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(Pgi) = (d*Pgi^3 + c*Pgi^2 + b*Pgi + a) * (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.

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.

MW Control Grouping

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.

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.

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.

Voltage Control Grouping

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.

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.

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 (page 92) 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 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 display filter status, select the **All generators in system** option.

Finally, to assign participation factors only for generators with valid display filters, select the **All generators with valid display 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.

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

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

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 cost of generation. Such curves can be defined using generator cost data files.

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 display 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.

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.

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.

Display Size

Size of the load.

Pixel Thickness

Thickness of the display object in pixels.

X and Y Location

X/Y location of the upper left corner of the display object.

Orientation

Specifies the direction to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See Anchored Objects (page 49) for details.

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 (page 94) for more details. The load multiplier value cannot be changed on this dialog.

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:

MW   =   LoadMultiplier(t) * (SMW + IMW * V + ZMW * V * V)

Mvar   =   LoadMultiplier(t) * (SMvar + IMvar * V + 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

LoadMultiplier(t) = (Case Load Multiplier) * (Area Schedule Value) * (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/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 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.

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.

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. You can toggle between limit sets using the *Limits Tab* of the PowerWorld Simulator Options Dialog, or the *Change Limit Set* button on the Line and Transformer Limit Violations Display.

Status

Current status of the device.

Pixel Thickness (Edit Mode only)

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 (page 49) 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).

View/Modify Line Shunts

Select to view the Transmission Line Shunt Information Dialog. This dialog is used to change the value of the line shunts. Line shunts are entered in per unit, with a positive B corresponding to capacitors and a negative B corresponding to reactors.

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.

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.

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.

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.

<div align="center">

Line Parameters

</div>

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.

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 (Rcomp = 0), or the rectifier voltage is specified (Rcomp = dc line resistance).

Status

Initial status of the dc line.

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

Per unit commuting resistance and reactance for the transformer.

Minimum, Maximum, and Actual Firing Angle

Minimum, maximum, and actual values of the firing angle for the rectifier.

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

Per unit commuting resistance and reactance for the transformer.

Minimum, Maximum, and Actual Firing Angle

Minimum, maximum, and actual values of the firing angle for the rectifier.

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.

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:

AC Line MW Flow	MW flow into the line or transformer at the near bus
AC Line Mvar Flow	Mvar flow into the line or transformer at the near bus
AC Line MVA Flow	Magnitude of MVA flow into the line or transformer at the near bus
AC Line Amp Flow	Magnitude of Amp flow (in amps) into the line at the near bus
AC Line MW Losses	Real power losses on the line or transformer in MW
AC Line Mvar Losses	Reactive power losses on the line or transformer in Mvar
DC Line MW Flow	MW flow into the dc line at the near bus
DC Line Mvar Flow	Mvar flow into the dc line at the near bus
MVA Limit	MVA limit for the line or transformer
DC Line Setpoint	Set point value for the dc line; see DC Transmission Line dialog (page 98)
Select a Field	Choose from over 80 different fields

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

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 non-phase shifting transformers, the phase shift angle should be set to zero. For phase-shifting transformers this value normally ranges from about -40° to 40°.

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 Line/Transformer Dialog.

Transformer Voltage Control

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). Note that automatic control is possible only if a regulated bus has been specified. These values can be seen by clicking on the **Automatic Control Options** button. Note that the **Tap Sensitivity** field provides an estimate of the change in the controlled bus voltage if the tap is increased by the step size value.

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.

Phase Shifter Control

A phase-shifting transformer can be placed on automatic MW control by checking the **Automatic Control Active** box. When it is on automatic 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 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.

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.

Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

Step Size (Degrees)

Phase shift change per step in degrees.

MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the Transformer Impedance Correction Dialog. To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the Line/Transformer Dialog.

View Transformer Correction Table or **Insert Transformer Correction Table**

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the Transformer Impedance Correction Dialog. Note that the table must prescribe at least two points in order to be defined.

Transformer Field Information

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.

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.

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

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.

Display Size

Size of the switched shunt.

Pixel Thickness

Thickness of the display object in pixels.

X and Y Location

X/Y location of the upper left corner of the display object.

Orientation

Specifies the direction in which to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See Anchored Objects (page 49) for details.

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.

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.

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) (page 214) 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.

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.

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.

Current Flow

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

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.

Element Identifiers

Choose whether to show the interface descriptions using the interface names or the interface numbers.

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.

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.

Chapter 7

Solving and Simulating a Case

This chapter describes the Run Mode options menu. This menu is used to customize and control the time domain and single Power Flow Solutions in the Run Mode.

The following material is included:

- PowerWorld Solution/Environment Options

- Controlling Simulations and Solving the Power Flow

- Description of Area Control and Transaction Modeling

- Equivalents

- Charts

- Contouring

- Sensitivities

- Scaling

- Bus View Display

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.

This section discusses the use of the PowerWorld Simulator Options Dialog by describing the options it presents.

<div align="center">

Power Flow Solution Options

</div>

The Power Flow Solution Tab offers various options regarding how Simulator solves the power flow problem. There are three sub-categories on the Power Flow Solutions tab. 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.

<div align="center">

Power Flow Solution: General

</div>

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.

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.

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.

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.

Bus Loss Sensitivities

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.

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.

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 (page 203) 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.

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 oneline 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 oneline diagram will become a dark shade of gray, and a message box will appear to announce that the system has experienced a blackout. To disable this behavior, select the Disable Showing Blackouts box. Representing the failure to converge as a blackout can be very helpful for presentation purposes. Very often, the power flow's failure to converge can be traced to the system's inability to serve the load demand, a situation that requires that load be "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 Oneline Hints

If checked, a hint will pop up when the user places the cursor over certain types of elements on the oneline diagram. By default, this option is not selected.

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.

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** tree 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.

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 Oneline 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 Oneline 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 Oneline File** to have Simulator open the oneline identified in the **Default Oneline 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.

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.

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 Oneline Templates (page 39).

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

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 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 (page 94) 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.

Percent Loading to Report Branch Limits

The percent thermal loading at which a branch is flagged as a violation.

Per Unit Low Voltage Limit, Per Unit High Voltage Limit

Acceptable per-unit limits for bus voltage. Any bus voltages that fall outside this range are flagged as violations.

Treat Transmission Line Limits as Equivalent Amps

Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If this box is checked, line limits are expressed in terms of their amp loading. Otherwise, line limits are checked in terms of MVA. See Line and Transformer Limit Violations (page 175) and see Limit Violations List (page 174) for details.

Line and Transformer Limit Set

Simulator allows the use of up to three limit sets for transmission lines and transformers. Select the desired limit set. Up to three limits for each line/transformer may be set on the Line/Transformer Dialog.

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 (page 119) for more information.

The following tasks are available from the Simulation Control menu and the Program Palette:

Starting the 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, Reset** command, or by clicking the reset button on the Run Mode Palette.

Pausing the 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 Paused 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.

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 Solution and Environment Options Dialog to initialize every solution from a flat start.

Solving the Power Flow

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.

Display Only

The **Display Only** option allows you to animate the oneline diagrams without solving the power flow equations. When Simulator is in Display Only mode, the animated flows always reflect the state of the system as of the last Power Flow Solution. Any changes you make to the system, such as switching lines in and out of service or changing a generator's output, are not reflected in the animated flows until you re-solve the system. In other words, in Display Only mode, Simulator will not solve the power flow equations automatically at each time step. The advantage of the Display Only mode is faster display animation. Thus, if you simply want to animate flows for a large system without modifying the system too frequently, you will find the Display Only mode useful.

To enable/disable the Display Only Option

- From the main menu select **Options/Tools, Solution/Environment**. This displays the PowerWorld Options Dialog.

- Check the **Display Only** checkbox on *Environment Tab* to enable the Display Only option. Leave the box unchecked to force Simulator to re-solve the power flow equations at each iteration.

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

$$ACE = P_{actual} - P_{scheduled} + \text{(a term dependent upon system frequency)}$$

where P_{actual} is the actual amount of MW flowing out of an area. If power is actually flowing into the area, P_{actual} is negative. $P_{scheduled}$ 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 Δf 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, $P_{scheduled}$ 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 four 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.

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

$$\lambda_i \ = \ IC_i \,(P_{gi}) \ = \ (\ b_i \ + \ 2c_i\, P_{gi} \ + 3d_i\, (P_{gi})^2\,) * \text{fuelcost} \qquad \text{\$/MWH}$$

The plot of $IC_i(P_{gi})$ as a function of P_{gi} is know as the *incremental-cost curve*. The economic dispatch for a system occurs when the incremental costs for all the generators (λ_i) are equal. This value is known as the system λ (lambda) or system incremental cost. Its value tells you how much it would cost to generate one more MW for one hour. The system lambda becomes important when trying to determine whether or not an area should buy or sell power. For example, if an area can buy power for cheaper than it can generate it, it might be a good idea for the area to buy power.

Area 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 (page 162) for a description of how to do this.

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

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 (page 122).

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 (page 196) 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 (page 122). This task has the following options:

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.

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 (page 120). 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).

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.

Import From/Export To

When contouring Area/Zone ATC values, these options become available. Check one of these two options for the desired ATC contour, and then select the area of interest from the drop down menu.

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.

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.

Action

Choosing Draw Contours will show the contour on the oneline diagram when you select either OK or Recalculate Contours. Remove Contours will remove any previously drawn contours from the display.

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:

Absolute Minimum:	The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.
Limit Minimum:	This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.
Nominal :	This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.
Limit Maximum:	This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.
Absolute Maximum:	The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Note: a representation of the color map is shown to the right of the values.

Ignore Above Min %

Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image.

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.

Import From/Export To

When contouring Area/Zone ATC values, these options become available. Check one of these two options for the desired ATC contour, and then select the area of interest from the drop down menu.

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.

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 to towards the Nominal Value. The Begin Fade Percentage specifies when the value starts to fade as a percentage of the largest distance for which this data point influences the contour.

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.

Default Color

For parts of the contour image which are not influenced by any of the data points, this color will be used.

Number of Points

This value sets the total number of color points used to define the color map. The color points are then set using the Colored Rectangles and Percentage Values. To change the color of a one of the rectangles, click on the rectangle.

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.

Sensitivities: Flows and Voltages

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 Display 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.

Sensitivities: Losses

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 display filters. The Bus Marginal Loss Sensitivities Table is a type of Case Information Display and thus exhibits features and behavior similar to all other case information displays. It has a local menu from which you can choose to find out more about a particular bus. You can sort records by any of the listed fields by clicking on the column headings. The table contains the following fields:

Number, Name

Number and name of the bus.

Area Number, Area Name

Number and name of the bus' area.

MW Sens.

Sensitivity of the loss function to an increase in the real power injection (generated power assumed positive) at the bus.

MVR Sens.

Sensitivity of the loss function to an increase in the reactive power injection (generated power assumed positive) at the bus.

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.

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 any combinations of areas, zones, and super areas. Use the *seller type* and *buyer type* options to indicate the type of the selling and purchasing entities. (The *Custom* option is not currently supported.)

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.

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%.

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.

Automatically Update

If checked, the PTDFs are automatically updated every time the power flow is solved.

Use Display 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 display filter set.

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 (page 135) for more information.

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.

Show Results For

PTDF results may be shown for branches, interfaces, areas, zones, or generators. Your selection for this option defines the contents of the results table at the bottom of the dialog.

PTDF Calculation Method

PTDFs may be calculated using either the full power flow Jacobian or only a portion of it. If you select the Full 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. 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. The lossless dc approximation is more efficient to compute, but the Full AC Approximation tends to be more accurate because it accounts for the relationship between reactive power flow and voltage magnitudes.

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.

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 Oneline Display Options Dialog.

Results Table

The Results Table occupies the bottom of the PTDF Dialog. It is a case-information display and thus shares many characteristics common to all other case information displays. Depending upon the value of the *Show Results for* option, the table will show results for lines/transformers, interfaces, areas, zone, or buses. The table features 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.

The contents of the table depend on what types of elements you have chosen to display, as described here:

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.

Interfaces

Shows the transaction distribution factors for the interface records. The following fields are shown:

Interface Name

Alphanumeric 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.

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 Increase

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.

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 Dialog

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.

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. 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 Table** lists the name and number of all 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** and **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.

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

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.

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 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 display.

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 Display Filters

- Power Flow and Quick Power Flow List Displays

- Limit Violations Displays

- Displays for All Major Case Components

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
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.

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.

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 display 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 display 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 display 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 display filter is not set, the display 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 Display 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.

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 Display Filters and Limit Violations displays, also fall into the category of case information displays. The next few topics discuss characteristics common to all case information displays.

Configuring the Case Information Displays

The content and format of all case information displays can be controlled through the Case Information Display Options Dialog. This dialog can be viewed by clicking Display/Column Options from the case information display's local menu.

To ensure that data presented by the case information displays always reflects the current state of the system, check the box labeled *Automatically Refresh*. 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.

To restrict the case information display's record set to cover only those areas and zones specified by the display filters, check the *Use Display Filters* box. If this box is unchecked, all records, regardless of zone or area, will be displayed. (The display filters option obviously does not apply to the Display Filters information display).

The next few options control the appearance of the case information display. Adjust the value labeled *Row Height* to define the height of the rows of the case information display. Click *View/Modify Custom Font* to define the font with which to display the records. Changing either the Row Height or Custom Font properties will cause the *Use Custom Font/Row Height* box to be checked. If you wish not to apply your custom specifications to the active case information display, uncheck the *Use Custom Font/Row Height* 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. You may also redefine the default settings to match your custom settings by clicking the *Set Font/Row Height as Default* button. Finally, click the *Auto Size All Column Widths* button to constrain all field widths to contain the widest data elements in each column.

The contents and visibility of each column in the case information display are defined using the next few controls. Select the column whose properties you wish to inspect or to modify in the box labeled *Column Number*. Change the field corresponding to that column by selecting a different field type from the drop-down list labeled *Column Field*. Adjust the physical width of the column by entering a new value in the *Column Width* text box, or change the column's numerical format using the *Total Digits* and *Decimal Places* inputs. You may also delete the column or insert a new column ahead of it by clicking the *Delete Column* and *Insert New Column* buttons, respectively. Note that, when you choose to insert a new column, the new column is defined with the same

properties, including field association, as the pre-existing column it now precedes. Therefore, you will want to change the newly inserted column's properties as described in this paragraph.

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.

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 (page 77) 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 (page 77) for more details.

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 (page 77) 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 (page 229).

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 (page 146) for more details.

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:

Standard Fields	Fields that cannot be modified directly from the case information display are colored black by default.
Enterable Fields	Fields that can be modified are colored navy blue by default. When selected, a cell containing an enterable field will display a tiny filled square in its bottom right corner. This box is called a cell handle.
Toggleable Fields	Fields whose values can be toggled are colored green by default. The values contained in toggleable fields are modified by left-clicking on them. Like cells in enterable fields, cells in toggleable fields display a cell handle when selected.

Besides indicating field type, color coding is also used to highlight violations of branch flow, generator MW or MVR output, and bus voltage constraints. Fields that are either at a limit or violating a limit are colored red by default. This, too, is configurable from the Case Information Displays tab on 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

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.

In the identifier fields, you can either directly enter the object's number or name, or use the associated combo box to select the identifier from a list of all identifiers of that type for the system. With the Find Line Dialog, once you have entered the **From Bus**, the **To Bus** combo box will only list those buses connected to the From Bus. Entering the **Circuit** value is optional; if it is omitted, the dialog returns with the first line found between the two buses.

Once you have selected the object's identifier(s), use the **Find** button to locate the object, or select **Cancel**. If **Find** is selected, the Case Information display is repositioned to display the requested

object. If the object is in an area and/or zone with a display filter set to "No", the display filter is automatically set to "Yes."

Display Filters

The Display Filters Display, invoked by clicking **Case Information, Display 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 display 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 Display 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 filters display's only enterable field is the one entitled *Display Filters*, which may assume only the values Yes and No. For example, if the Display Filters setting for an area is *No*, then any case information display configured to enforce display 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 Display Filters 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 Display Filter records.

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 (page 120)

for more details. You can toggle the value of the area's AGC Status (except in Viewer) by left clicking on the entry. Valid entries in this field include:

- Off AGC

  Area is off AGC. Generation must be adjusted manually to meet changes in load and losses. If it is not, the system slack will be forced to pick up the balance.

- Part AGC

  Area is on AGC, with generation dispatch controlled by its units' participation factors.

- ED

  Area is on economic dispatch control so that generation is dispatched in order of cost.

- OPF (only used with Simulator OPF)

  Area is on OPF control. 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.

Scheduled 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 (page 121) for a description of the difference between base and scheduled transactions.

Interchange 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

Area control error in MW.

Lambda

The area's marginal cost.

Loss (MW)

Total real power losses for the area.

Report Limits

Determines whether limit violations for the area are reported in the Limit Violations List. Click on this field to toggle its value. Click on the "Toggle All Yes" or "Toggle All No" local menu items to set the value of entries in this column.

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.

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 (page 212)); 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 (page 101) 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.

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 buses. 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 display 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.

MW Load, Mvar Load

Total real and reactive load at the bus. If no loads are located at the bus, these fields are blank.

MW Gen, Mvar Gen

Total real and reactive generation at the bus. If no generators are located at the bus, these fields are blank.

Loss MW Sensitivity

A measure of the sensitivity of system losses to power injection at the bus.

DC Line 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 display 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 Bus #, 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 Oneline from the local menu to view additional information about the rectifier bus.

Inv Bus #, 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 Oneline 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.

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 display 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'.

MW, 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 (page 120) 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 (page 120) 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 properties.

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 Fuel 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 display 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.

ID

Single character ID used to distinguish multiple generators at the same bus; '1' by default.

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 (page 93) for details. Please note that these values can be saved/loaded using the Generator Cost Data auxiliary file type.

Cost

Operating cost for the generator in $/hr.

IC

Incremental cost to produce an additional MWh. This can be expressed as $dC(Pgi)/dPgi$, where C denotes the generator's cost of operation in $/hr and Pgi 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 $\partial Ploss/\partial Pgi$. 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 $1/(1 + \partial Ploss/\partial Pgi)$. 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).

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 are 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 Interface 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 display 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 (page 40) for details.

The Interface display contains the following fields:

Name

Alphanumeric identifier for the interface (24 characters maximum).

MW Limit A, Limit B, Limit C

Ratings for the interface in MW. Simulator allows the use of three different limits sets for interfaces. The limit set used here is the same as the limit set for transmission lines and transformers. You can toggle between limit sets using either the Limits tab of the PowerWorld Simulator Options Display, the Line and Transformer Limit Violations Display, or the Line/Transformer Display.

Flow

Current MW flow on the interface.

Percent

Percentage loading of the interface, calculated as Flow/(MW Limit) * 100.

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 display 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'.

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

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

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

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

See Transformer Display (page 167) 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.

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 display 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 (page 208) and see Load Modeling (page 94) for more information.

SMW, SMVAR, IMW, IMVR, ZMW, ZMVR

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 (page 94) for more details on how bus load is modeled.

Load Variation Schedule

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 display 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.

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 (page 94) 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**.

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 (page 97) 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 display 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*.

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 (page 121) 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 display 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.

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*.

Buy Price ($/MWhr)

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

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.

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.

OK

Click to save any changes and close the dialog.

Cancel

Click to close the dialog without saving any changes.

Delete

Click to delete the current transaction and close the dialog.

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.**

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 Oneline. 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 display 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.

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.

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 Oneline.   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 display 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.

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.

Actual 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.

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.

Type

Type of transformer. Possible values include

- **Fixed**

  The tap positions are fixed

- **LTC**

  The tap ratio changes to regulate bus voltage

- **Phase**

  The phase angle changes to regulate real power flow.

Tap/Phase

Indicates the tap ratio for LTC and fixed transformers and the phase shift angle in degrees for phase-shifting transformers.

XF Auto

If the value of this field is *Yes*, the transformer will automatically change its tap or phase angle to keep the regulated value within the specified regulation range, provided that the *Auto XF* field of its associated area is set to *Yes* and transformer tap/phase control has not been disabled for the entire case. The *Auto XF* field of individual areas is set from the Area Display, and case-wide transformer control can be set from the Power Flow Solution tab of the 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 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.

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.

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.

Transformer Field Information

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.

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.

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

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.

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.

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:

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.

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 (page 120) for details on these types of generation control.

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.

New Area

Use the dropdown box to select an area to add to the super area. Click the **Add New** to add the selected area to the super area.

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.

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.

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 Num

Zone's number, between 1 and 999.

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.

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 Oneline. 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.

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 Number, Slack Name

Number and name 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.

Load MW, Load Mvar

Total real and reactive load for the island.

Gen MW, Gen Mvar

Total real and reactive generation for the island.

Scheduled Export

The power scheduled to be provided by 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.

Limit Violations

Selecting **Case Information, Limit Violations** from the main menu yields a submenu of displays that show limit violations. These displays cover:

Bus Voltage Violations

Line/Transformer Violations

Interface Violations.

By default, the violations shown on all these displays pertain only to devices that reside in areas having **Report Limits** set to "Yes." This field is set on the Area Records Display. You can also override this option and show violations for the entire case.

Bus Voltage Magnitude Limit Violations

The Bus Voltage Magnitude Limit Violations display identifies buses whose voltage magnitude lies outside a user-specified threshold. Entries are sorted by the PU Volt field (per unit voltage magnitude) when the display first appears. There are several options on this display that control what violations are reported. The following fields and controls are shown at the top of the display:

Number of Line/Transformer Violations

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

Number of Bus Voltage Magnitude Violations

Shows the total number of violations reported on this, the Bus Voltage Magnitude Limit Violations, display. This field is read-only.

Number of Interface Violations

Identifies the total number of interface violations. These violations are shown on the Interface Violations Display. This field is read-only.

Use Area Limit Filters

If this option is checked, then limit violations are shown only for devices that are in areas with Report Limits set to "Yes." (This field is set on the Area Records Display). If the option is not checked, then violations are reported for all areas in the case. When you toggle this field, the display is automatically updated.

Automatic Refresh

Selecting the Automatic Refresh option forces the Limit Violations Display to be refreshed whenever a new Power Flow Solution is available. For large cases, you may wish to leave this option unchecked to avoid performance problems. To refresh the display manually, press the **Refresh** button.

Minimum PU Voltage, Maximum PU voltage

Controls the values for reporting of per unit bus voltage magnitude violations. Buses with a voltage magnitude below the Minimum PU Voltage field or greater than the Maximum PU Voltage field are reported as violations. The display is automatically updated when you press the enter key in this field. This field is available on the limits tab of the PowerWorld Simulator Options Dialog, as well.

A table identifying all bus voltage violations that meet the above criteria consumes the rest of the dialog. Since this table is yet 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 voltage violations, copy them to the Windows clipboard for use with another application, modify the format and content of the violations listing, and view the Information Dialog, Quick Power Flow List, or Bus View Display corresponding to the violating bus. There are four commands available at the top of the list.

The list contains the following fields by default:

Number, Name

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

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.

Line and Transformer Limit Violations

The Line and Transformer Limit Violations Display shows selected transmission lines and transformers that have exceeded a user specified threshold. When the display first appears, entries are sorted by the Percent field, which indicates the severity of the violation. The Line and Transformer Limit Violations Display offers several options that control what violations are reported. The following fields are shown at the top of 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.

Use Area Limit Filters

If this option is checked, then limit violations are shown only for devices that are in areas with Report Limits set to "Yes." (This field is set on the Area Records Display). If the option is not checked, then violations are reported for all areas in the case. When you toggle this field, the display is automatically updated.

Automatic Refresh

Selecting the Automatic Refresh option forces the Limit Violations Display to be refreshed whenever a new Power Flow Solution is available. For large cases, you may wish to leave this option unchecked to avoid performance problems. To refresh the display manually, press the **Refresh** button.

Percentage Threshold for Reporting

Controls the threshold for reporting of branch flow limit violations. All transmission lines and transformers with an MVA loading at either end of the device greater than this threshold percentage of their MVA limit are reported. The display is automatically updated when you press the enter key in this field.

Treat Transmission Line Limits as Equivalent Amps

Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the *Treat Transmission Line Limits As Equivalent Amps* box 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

$$Amp\ Rating = \frac{MVA\ Rating}{\sqrt{3}*Base\ kV}*1000.$$

Line/Transformer Limit Set

Shows the current limit set being used for checking for violations. Simulator can recognize up to three limits for each transmission element, which are commonly referred to as rates A, B, and C.

Change Limit Set

Click on this button to change the limit set. The display is automatically refreshed using the new limit set.

A table identifying all branch flow violations that meet the above criteria consumes the rest of the dialog. Since this table is yet 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 branch flow 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 line or transformer, and view the Information Dialog, the Quick Power Flow List, or Bus View Display corresponding to either the *from* or *to* terminal of the branch.

The branch flow limits display contains 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 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.

From Flow, To Flow

If the *Treat Transmission Line Limits As Equivalent Amps* option is selected, then the *From Flow* and *To Flow* fields indicate MVA loading for transformers and amp loading for lines at 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.

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 flow/limit * 100.

Units

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

Interface Violations

The Interface Violations Display shows monitored interfaces whose flow has exceeded a user-specified threshold. You can view all defined interfaces using the Interface Records Display. Entries are sorted by the Percent field when the Interface Violations Display first appears so that the list of violations is arranged by severity.

The display presents several options for controlling and monitoring which violations are reported. These controls reside at the top of the Interface Violations Dialog and include the following:

Number of Line/Transformer Violations

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

Number of Bus Voltage Magnitude Violations

Shows the total number of violations reported on the Bus Voltage Magnitude Limit Violations Display. This field is read-only.

Number of Interface Violations

Shows the total number of violations reported on this, the Interface Violations Display. This field is read-only.

Use Area Limit Filters

If this option is checked, then limit violations are shown only for devices that are in areas with Report Limits set to "Yes." (This field is set on the Area Records Display). If the option is not

checked, then violations are reported for all areas in the case. When you toggle this field, the display is automatically updated.

Automatic Refresh

Selecting the Automatic Refresh option forces the limit violations display to be refreshed whenever a new Power Flow Solution is available. For large cases, you may wish to leave this option unchecked to avoid performance problems. To refresh the display manually, press the **Refresh** button.

Percentage Threshold for Reporting

Controls the threshold for reporting interface limit violations. All interfaces with a MW loading greater than this threshold percentage of their MW limits are reported. The display is automatically updated when you press the enter key in this field.

A table identifying all interface flow violations that meet the above criteria consumes the rest of the dialog. Since this table is yet 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 interface flow violations, copy them to the Windows clipboard for use with another application, modify the format and content of the violations listing, and view the information dialog of the violated interface.

The interface flow limits violation display contains the following fields by default:

Name

Alphanumeric identifier for the interface (24 characters maximum).

MW Flow

Current MW flow on the interface.

MW Limit

Interface limit in MW.

Percent

Percentage loading of the interface, calculated as MW Flow/MW Limit * 100.

Elements

Number of elements in the interface.

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 Oneline Display Options (page 48) 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.

Ownership

The Ownership feature is new to Simulator version 6.0. Buses, loads, generators and lines may now 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 (page 180).

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. When the selection to save as a PTI version 24/25 format is made, Simulator detects whether or not any owner data exists. If it does exist, then Simulator will notify you that owner data exists, and prompts you to continue and save as a version 25 file. If no is selected, a warning is displayed that the owner data will not be saved, and then yes can be selected to save as a version 24 format file, otherwise the raw file will be saved as version 25 format. If no owner data is detected, Simulator automatically saves as version 24 format.

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.

Chapter 9

Injection Groups

Injection groups are a new feature to PowerWorld Simulator. 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.

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.

Participation Points

The right half 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.

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

- SPECIFIED: The participation factor was specified as a constant.

- MAX GEN INC: The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

- MAX GEN DEC: The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

- MAX GEN MW: The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

- SPECIFIED: The participation factor was specified as a constant.

- LOAD MW: The participation factor was defined as the size of the load in MW.

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 display 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
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:

- Bus Palette

- Appending Additional Power Flow Components to the Existing Case

- Automatically Insert Transmission Lines, Generators, Loads, and Interfaces on the Oneline

- Renumber the Buses in a Case and/or Oneline

- Drawing Options

- Scale the Case

Bus Palette Overview

The **Bus Palette** is designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. It lists for you the buses that you have already added to the oneline, the buses that have not yet been drawn that neighbor those buses, and the set of all buses that have not yet been added to the drawing. By selecting and dragging a bus name from either the **Undrawn Neighbors** list or the **All Undrawn Buses** list to a location on the oneline diagram where you would like that bus to appear, you can add that bus to the drawing with very little effort. The Bus Palette is especially useful for adding a region of an interconnection in its entirety.

To display the Bus Palette, select **Insert, Show Bus Palette** from the main menu.

Using the Bus Palette

The Bus Palette is a tool designed to accelerate the building of a oneline diagram based on a pre-existing power flow case. To display it, select **Insert, Show Bus Palette** from the main menu.

The Bus Palette features the following controls:

Displayed Buses

Lists those buses defined in the power flow case that have already been added to the oneline diagram. When you click on an entry in the Displayed Buses List, the contents of the **Undrawn Neighbors List** will be refreshed to identify all buses that neighbor the bus you selected that have not yet been added to the oneline.

Double click on an entry in this list to pan to it so that the selected bus appears in the center of the screen. Right-click on an entry and choose either *Quick Power Flow List* or *Bus View Display* to learn more about the selected bus.

Undrawn Neighbors

Lists those buses defined in the power flow case that neighbor the bus selected in the **Displayed Buses List** that have not yet been drawn. To add the undrawn bus 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 bus. When you let go of the left mouse button, the **Bus Information Dialog** box for that bus will appear. Use the Bus Information Dialog to change display parameters for the bus (such as its orientation and size) and click **OK** to finish dropping the bus onto the oneline. The bus you have just added will be appended to the end of the Displayed Buses List and will also be added to the **History List** so that you can identify its own undrawn neighbors quickly.

Right-click on an entry in this list and choose either *Quick Power Flow List* or *Bus View Display* to learn more about the selected bus.

All Undisplayed Buses

Lists those buses defined in the power flow case that have not yet been added to the oneline diagram, regardless of whether the bus neighbors a displayed bus or not. This list functions identically to the **Undrawn Neighbors List**. To add the undrawn bus 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 bus. When you let go of the left mouse button, the **Bus Information Dialog** box for that bus will appear. Use the Bus Information Dialog to change display parameters for the bus (such as its orientation and size) and click **OK** to finish dropping the bus onto the oneline. The bus you have just added will be appended to the end of the Displayed Buses List and will also be added to the **History List** so that you can identify its undrawn neighbors quickly.

Right-click on an entry in this list and choose either *Quick Power Flow List* or *Bus View Display* to learn more about the selected bus.

History List

Identifies the buses you have added to the oneline diagram using the bus palette since the last time you opened the bus palette. Select a name from this list to display the undrawn neighbors of the corresponding bus.

Click **Close** when you are done using the Bus Palette.

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.

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 (page 57). 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 mode, in the Edit Mode select **Options/Tools, Auto Insert, Lines** from the main menu.

The dialog has the following options:

Line Spacing on Vertical and Horizontal Buses

Specifies the spacing in screen coordinates between adjacent automatically inserted lines emerging from a terminal bus. Separate spacing distances can be specified for vertical and horizontal terminal buses.

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.

Insert Equivalenced Objects

If this option is checked, equivalenced objects modeled as lines will also be automatically inserted with the real transmission lines.

Insert Transmission Line Stubs

Check this option to force Simulator to 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 Length**, and a third segment joining the two stubs.

Insert Circuit Breakers

If this option is checked, circuit breakers are automatically added at each end of the new lines.

Insert Pie Charts

If this option is checked, a line pie chart is automatically added at the middle of each of the new lines.

Use Only Selected Buses

Instead of having Simulator automatically insert line and transformer display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Insert Text Fields

If this box is checked, the text fields defined in the original oneline will be displayed.

Bus Tie Breaks

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 lines will physically show up on the oneline or not. The second allows only one circuit breaker to show up on the oneline if you choose to show the stubs. This could be useful for determining real lines from bus tie breaker.

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 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 brand new generator, see

Generator Display Objects (page 54). 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 buses for the end of 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 mode, in the Edit Mode select **Options/Tools, Auto Insert, Generators** from the main menu.

The dialog has the following options:

Line Spacing on Vertical and Horizontal Buses

Specifies the spacing in screen coordinates between adjacent automatically inserted generators emerging from a terminal bus. Separate spacing distances can be specified for vertical and horizontal terminal buses.

Minimum kV Level

Simulator will automatically draw generator display objects between terminals 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.

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.

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 (page 55). 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 buses for the end of 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 mode, in the Edit Mode select **Options/Tools, Auto Insert, Loads** from the main menu.

The dialog has the following options:

Line Spacing on Vertical and Horizontal Buses

Specifies the spacing in screen coordinates between adjacent automatically inserted loads emerging from a terminal bus. Separate spacing distances can be specified for vertical and horizontal terminal buses.

Minimum kV Level

Simulator will automatically draw load display objects between terminals 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.

Insert Equivalenced Objects

If this option is checked, equivalenced objects modeled as loads will also be automatically inserted with the real transmission lines.

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.

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 display 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.

<p style="text-align:center;">**Automatic Setup of Bus List Options**</p>

This section of the display 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 Display Filters

Same as the **Load All Buses In Case** option except only those buses with valid display 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.

- 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.

<div align="center">**Bus Change Options**</div>

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.

Setting Default Drawing Options

The Default Drawing Options Dialog is used to define the default sizes of display objects, as well as various other display parameters. Many of these options are updated dynamically when the user enters a non-default value for a new display object. 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.

For reference, when the zoom level is at its nominal value (100%), the size of the screen is 100 by 100.

Bus Size

Indicates the default size for new bus display objects.

Bus Pixel Thickness

Specifies the default pixel thickness used in drawing new bus display objects.

Load Size

Indicates the default size for new load display objects.

Generator Size

Indicates the default size for new generator display objects.

Switched Shunt Size

Specifies the default size for new switched shunt display objects.

Bus Device Pixel Thickness, Bus Device Color

Bus devices include loads, generators, and switched shunts. The Bus Device Pixel Thickness and Bus Device Color fields control the pixel thickness and color with which such objects are drawn. Use the **Change Device Color** button to change the device color, or double-click on the color field itself.

Circuit Breaker Size

Defines the default size for new circuit breaker display objects.

Pie Chart Size

Specifies the default size for new pie chart display objects.

Line Pixel Thickness, Line Color

Indicates the pixel thickness and color with which new branches (transmission lines and transformers) are drawn. To change the line color, either click the **Change Line Color** button, or double-click on the color field itself.

Transformer Symbol Size

Indicates the default size for new transformer symbol objects.

Use Zoom Levels

If checked, new objects are set so they are conditionally displayed on the oneline based upon their zoom level. The objects are shown as long as the zoom level is between the **Low Zoom** value and the **High Zoom** value.

Low Zoom

Lowest zoom value at which a new object is shown on the oneline, as long as the **Use Zoom Level** field is checked.

High Zoom

Highest zoom value at which a new object is shown on the oneline, as long as the **Use Zoom Level** field is checked.

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.

Automatically Insert CBs on New Lines/Transformers

Check this box to command Simulator to add circuit breakers at each end of new transmission line and transformer display objects.

Set Auto Field Insertion

See Inserting and Placing Multiple Display Fields (page 77) for more information on this topic.

Set kV Color Coding

See kV-Based Color Coding (page 195) for more information on this topic.

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.

Add to External System

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.

Add to Study System

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.

Voltage Levels to Include

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 Oneline to Object** to locate and select the unlinked object on the oneline 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 oneline diagram. The **Oneline 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 oneline 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 oneline. Choose **Show Dialog** to show the object's corresponding information dialog. Finally, choose **Pan Oneline to Object** to locate and select the corresponding display object on the oneline.

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. 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 Table** lists the name and number of all 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** and **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.

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

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. Keep in mind that scaling generation is NOT the same as doing an area control dispatch. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control.

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

- Run Mode Display Object Information

- Transformer AVR Dialog

- Transformer Impedance Correction Tables

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.

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.

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. 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.

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 (page 150) 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 (page 172) 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 (page 203).

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 **Disconnect** opens all of the transmission lines incident to the bus, disconnecting the bus from the rest of the system. Selecting **Connect** 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.

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.

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.

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.

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.

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.

<div align="center">

MW and Voltage Control

</div>

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.

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.

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.

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 any time you manually change its output. You could then place the generator back on AGC control by using this dialog.

Voltage Control

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.

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.

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 (page 92) 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.

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 (page 93) 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 $C(Pgi) = (d*Pgi^3 + c*Pgi^2 + b*Pgi + a) * (fuel cost)$, 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.

Generator Economic Curves

Four characteristic curves describe the efficiency and resulting costs associated with operating a particular generating unit. These four curves plot

- Fuel Cost

- Heat Rate

- Input-Output

- Incremental Cost

Simulator can display plots of all these curves. To display a particular plot for a generator, right-click on the generator in Run Mode to display its local menu, and then select the plot you wish to see. The plot will be presented in its own window. The windows for all plots exhibit identical characteristics. For example, the current operating point is identified by a red filled circle. Right clicking on an open area of the window displays the plot's local menu which allows you to print the plot, save it to a file, or copy it to the clipboard for use in other programs. To adjust the length and number of intervals shown on an axis, right-click on the *axis* (not on the *numbers*) 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.

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.

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 (page 94) for more details. The load multiplier value cannot be changed on this dialog.

Line/Transformer Information (Run Mode)

The Line/Transformer Information dialog box is used to view information about each transmission line and transformer in the system. You may use this dialog also to change many of the properties of lines and transformers (except in Viewer).

The Run Mode version of this dialog is very similar in content to its Edit Mode counterpart.

The Line/Transformer dialog sports the following fields:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side.

To Bus Number and Name

To Bus number and name.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Find By Number

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

Find By Name

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

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.

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. You can toggle between limit sets using the *Limits Tab* of the PowerWorld Simulator Options Dialog or the *Change Limit Set* button on the Line and Transformer Limit Violations Display.

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 *View/Modify Line Shunts* 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.

View/Modify Line Shunts

Select to view the Transmission Line Shunt Information Dialog. This dialog is used to change the value of the line shunts. Line shunts are entered in per unit, with a positive B corresponding to capacitors and a negative B corresponding to reactors.

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.

Off-nominal Turns Ratio, Phase Shift Degrees

For transformers only, these fields show either the off-nominal turns ratio or the phase shift angle in degrees. See Transformer Modeling (page 101) for details.

Automatic Control Active

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 (page 150) 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 Transformer Control Enabled* and *Area Transformer Control Enabled* check boxes.

Case Transformer Control Enabled and Area Transformer Control Enabled

Both of these boxes must be check for a transformer to be controlled. **Case Transformer 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 either of these does NOT automatically make the entire case or entire area on control.

Automatic Control Options

Click this button to display either the Transformer AVR information Dialog for LTC transformers or the Transformer Phase Shifter Information dialog for phase shifting transformers.

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 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 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.

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).

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.

Chapter 12

Contingency Analysis

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 switching of transmission lines and transformers, the loss or recovery of a particular generating unit, or the shifting of load from one bus to another. Simulator uses a full Newton solution to analyze each contingency. While this approach is not as fast as those using approximate techniques such as injection methods and dc load flow, the results tend to be significantly more accurate, particularly in gauging voltage/var effects. Moreover, the performance of Simulator's contingency analysis tools should suffice for moderately sized systems. If your system is very large or you have a great many contingencies to analyze, we suggest that you may find it worthwhile to use a dedicated contingency analysis package to identify a subset of critical contingencies, and then use Simulator to provide exact analysis for that subset.

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 (page 222)), 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. 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 Monitoring Violations Tab implements a function similar to that of display filters in Simulator. It governs which areas and zones of the system will be monitored for violations in response to each contingency, and over what range in voltage levels. 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 Control/Results Tab chronicles the performance of the contingency analysis and allows you to produce a report detailing the results of the run. 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.

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.

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 on it.

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 (page 219)), 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 or interface violation. If there are no branch or interface 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.

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. To display one of these, click on the corresponding tab. The contingency definition display identifies the actions that comprise the contingency selected in the contingency records display. The contingency violations display lists all violations that resulted from the selected contingency.

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 (page 226) for more details.

You may choose to remove all contingencies from the current contingency list by clicking the **Clear All** button. Furthermore, you may close the Contingency Analysis Dialog at any time either by clicking **Close** or by selecting *Close* from the local menus of the Contingency Records Display or the Contingency Violations Display.

Monitoring Violations Tab

The Monitoring Violations Tab allows you to restrict the search for violations resulting from each contingency to by area, zone, and nominal voltage level. To have Simulator look for violations throughout the system, regardless of the violated element's associated area or zone or its nominal voltage, simply uncheck the **Use Area/Zone Limit Filters** box. To limit the search for violations to only a portion of the entire system, make sure that this box is checked.

When the Area/Zone Limit Filters box is checked, a two-tabbed notebook containing two **Monitored Region Information Displays**, one for areas and one for zones, will appear. The Monitored Region Information Displays identify each area and zone in the system, whether limit violations should be reported for that area and zone, and, if so, over what range in nominal kV level should violations be flagged. The Monitored Region Information Display is a class of Case Information Display and, as such, possesses properties and controls shared by all other case information displays. The following fields are shown by default in each display:

Number, Name

The number and name of the area or zone.

Report Limits

A toggleable field that indicates whether the search for violations should include elements that are located in that area or zone. To toggle the value of this field, simply double-click on it.

Report Min kV, Report Max kV

Indicates the minimum kV that an element must meet for it to be considered as a limit violation. Any elements in the area or zone that do not fall in the range of nominal voltage defined by these two values will not be flagged as a violation.

Note that the area and zone limit filters work together to exclude elements from the set of limit violations. In order for an element to be excluded from the limit violations search, both its area limit criteria and its zone limit criteria must be set appropriately. If a violated element qualifies for consideration through either its area membership or its zone membership, it will be reported as a 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 three-page notebook, each page of which concerns a different aspect of the contingency analysis. These tabs are:

- Identifying Violations

- Modeling

- Base Case

- Report Writing.

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: Identifying Violations

The Identify Violations Tab allows you to shape how limit violations are detected. It is divided into three groups of controls, each concerned with a particular type of power system element.

The first set of option controls concern branch thermal violations. The controls include:

Pre-Contingency Ratings

Governs which rating set to use in identifying violations that exist in the Base Case.

Post-Contingency Ratings

Governs which rating set to use in identifying violations that result from contingencies.

Treat Transmission Line Ratings As

Provides the option of identifying thermal overloads in terms of either the current (amps) or the complex power flowing through the branch. Gauging violations in terms of current is usually preferred because the physical mechanism for the limitation is actually conductor heating due to

current flow. MVA limits derived from amp limits by assuming a particular voltage level that may not reflect reality.

Enforce % of Thermal Rating

Allows you to apply a factor to the branch's rating so that limit violations are flagged at some other limiting value. For example, to incorporate TRM into their ATC methodologies, some regions apply a multiplier to their branches' ratings so that limit violations are flagged at some value lower than their physical limitation, thus achieving a conservative cushion for line flows. This option allows you to implement such a scheme in Simulator.

The next set of options govern the detection of bus voltage violations. Any bus voltage that falls outside the range defined by the settings for **Per-Unit High Voltage Limit** and **Per-Unit Low Voltage Limit** is identified as a voltage violation.

The next option, **% Loading to Enforce Interface Limits**, is conceptually equivalent to the **Enforce % of Thermal Rating** option, except that it pertains to the enforcement of interface limitations. Any interface whose flow exceeds the product of the % Loading to Enforce Interface Limits and the interface's defined limit will be flagged as an interface violation.

Finally, in reporting violations, you may choose to identify buses either by name or by number by choosing appropriately in the **Identify Buses By** option box.

Except for the pre- and post-contingency ratings settings, these options affect detection of both pre- and post-contingency limit violations.

Options Tab: Modeling

The Modeling Tab controls 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: Base Case

The Base Case Tab 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. The Base Case Tab allows 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 Control/Results Tab (page 221) 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 any of the nine options on this tab, you can include additional information in the report.

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. In other words, the report will mirror the contents of the Contingency Analysis Monitoring Violations Displays.

Base Case Outages

If this item is checked, the report will list the limit violations that existed in the Base Case.

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.

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.

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.

Control / Results Tab

The Control/Results 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, and enables you to produce a detailed report on the results of the contingency analysis. The top half of the Control/Results 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**.

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.

To set the case currently in memory as the base or reference case for subsequent contingency studies, click the **Set Reference** button. See Contingency Case References (page 227) for more information.

Automatically Generating a Contingency List

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 single-outage contingency list automatically. It contains the following controls:

Insert Single Line Contingencies

Check this box to include all single transmission outages in the automatically generated list.

Insert Single Generator Contingencies

Check this box to include all single generator outages in the automatically generated list.

Bus Identifiers and Name Prefixes

When generating the contingency definitions, Simulator will also attempt to assign unique names to each contingency. Each contingency name will identify the terminal bus or buses of the outaged element. The terminal buses will be identified either by name or by number according to the option you choose for the Bus Identifiers setting. Added to this identification will be a prefix that you specify. For branch contingencies, the prefix you supply for the *Single Line Name Prefix* option will be used. For generator contingencies, the prefix you supply for the *Single Gen Name Prefix* option will be used.

Only Insert for Lines/Gens With Display Filters

Check this box to limit the automatically generated contingency list to include only those elements that reside in areas or zones whose display filter setting is active. If you leave this box unchecked, all single elements in the case will be included in the contingency list.

Delete Existing Contingencies

Check this box to delete any contingencies that have already been defined before adding the new automatically generated contingencies. If you leave this box unchecked, the new contingencies will simply be appended to the list that already exists.

Press the **Insert Contingencies** button to generate the single-outage contingency list. Click **Cancel** to abort the automatic contingency creation.

Loading a Contingency List from a File

Simulator can load contingency definitions from a text file. The contingencies may be specified either in PowerWorld Contingency Format, or in the contingency format used for PSS/E Versions 23 through 25 (see PSS/E Contingency Format (page 223) for limitations). PowerWorld Contingency Files have the extension *.ctg* by default, whereas Simulator assumes that PSS/E-formatted contingencies are stored in files having the default extension *.con*.

To load contingencies from a text file, click the **Load From File** 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.

PowerWorld Contingency Format

PowerWorld has developed a special format for defining contingencies for Simulator's contingency analysis tool. Each contingency definition is defined within a block structure as follows:

EVENT *contingency_name*

    Action 1

    .

.
.

Action n

END

Thus, each contingency definition has a user-specified name *contingency_name* and consists of one or more actions bounded by the EVENT and END keyword lines. Each action specification generally begins with a keyword that identifies the type of element (BRANCH, GEN, LOAD, SHUNT) that is participating in the contingency, continues with an identification of the element or elements involved, and concludes with a verb (OPEN or CLOSE). (Note that load shift specifications deviate slightly from this format.) Here are examples of the contingency actions that Simulator recognizes:

Transmission Lines and Transformers:
    BRANCH *bus1#* *bus2#* *id* OPEN
    BRANCH *bus1#* *bus2#* *id* CLOSE

Generators
    GEN *bus#* *id* OPEN
    GEN *bus#* *id* CLOSE

Shunts
    SHUNT *bus#* OPEN
    SHUNT *bus#* CLOSE

Loads
    LOAD *bus#* *id* OPEN
    LOAD *bus#* *id* CLOSE
    LOAD *bus1#* MOVE_PQ_TO *bus2#* *x* MW
    LOAD *bus1#* MOVE_PQ_TO *bus2#* *x* PERCENT

Note that Simulator always assumes that elements are identified by bus number in these definitions. The formats should be fairly self-explanatory, except perhaps the one specifying a movement of load from one bus to another. In that case, *bus1#* identifies the number of the bus from which you are moving the load, *bus2#* identifies the number of the bus to which you are moving the load, and *x* specifies the quantity of load that is being moved in either MW or PERCENT. The MOVE_PQ_TO keyword suggests that both real and reactive load will be shifted from bus1 to bus2 such that the power factor is maintained constant.

PSS/E Contingency Format

Simulator can read and write parts of the contingency format used by Power Technology's PSS/E Versions 23 through 25. However, the current version of Simulator does not support the use of the SET, CHANGE, ALTER, MODIFY, REMOVE, INCREASE, or DECREASE keywords, and 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.

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 allows you to save contingencies using either the

PowerWorld Contingency Format or Power Technology's contingency format for PSS/E. To specify the format for the contingency file, set the *Save As Type* option accordingly.

Contingency Definition Display

The Contingency Definition Display is used to identify the actions that comprise the contingency that is currently selected in the Contingency Records Display. Click on a row of the display to view the Contingency Element Dialog for that particular action. The Contingency Element Dialog allows you to change the action's definition and even to remove the action from the contingency definition.

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. 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

Indicates the type of violation, or what violation was committed. The allowable values for this field are *Branch MVA*, *Branch Amp*, *Bus Low Volts*, *Bus High Volts*, and *Interface MW*.

Element

Names the element that suffered the violation.

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*.

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.

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 three ways:

- By pressing the **Insert New Element** button on the Contingency Case Dialog.

- By clicking on a contingency action in the Contingency Case Dialog's **Contingency Elements** table.

- By selecting a contingency action from the Contingency Definition Display.

The Contingency Element Dialog features the following controls:

Element Type

Indicates whether the element involved in the contingency action is a branch, a generator, a load, or a switched shunt.

Element Action

Defines the change specified by the contingency action. All element types may be opened or closed. Only loads may be moved.

Action Parameter

This control is enabled only for load shift contingencies. Use it to specify the quantity of load to be moved from one bus to another.

Action Parameter Units

This control is enabled only for load shift contingencies. Use it to specify whether the quantity of load expressed in the Action Parameter entry represents MW or Percent. In either case, it is assumed that both real and reactive load will be moved such that the power factor will remain constant.

Element Identifiers

Use these controls to identify the element involved in the contingency using its From Bus number and, if necessary, its To Bus number and ID.

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.

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.

Contingency Elements

Lists the actions that comprise the contingency. Click on an entry in this table to invoke the Contingency Element Dialog, which allows you to modify or delete the action.

Clear

Removes all actions from the contingency definition. The Contingency Elements Table will then appear blank, indicating that the contingency involves no associated actions.

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.

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 control, 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.

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.

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 reset the case in memory to correspond to the system state prior to applying the contingency.

- 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 Control/Results Tab.

To pause the contingency run once it has started, click either **Pause Run** on the Contingency Tab or **Pause** on the Control/Results Tab. To resume a paused contingency run, click the **Continue** buttons on either the Contingency Tab or the Control/Results Tab. Finally, to terminate a contingency run, click the **Abort** button on the Control/Results Tab. The status indicator will inform you of the run's current state.

To solve a single contingency, identify its violations, and then return to the pre-contingency system state, 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*.

To solve a single contingency and set it as the reference, or 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 (page 227) for more information.

Contingency Case References

The contingency analysis will always implement the contingency for the case that is currently in memory. During automatic processing of the contingency list, the pre-contingency case is always loaded into memory after a contingency is executed, so that all contingencies start from a common Base Case. When you solve contingencies one at a time instead, you can decide whether to restore the Base Case after solving the contingency by choosing between the *Solve Selected Contingency* and *Solve and Set As Reference* options from the Contingency Record Display's local menu. Alternatively, you may click the **Set Reference** button on the Contingency Analysis Control/Results Tab. Furthermore, whenever the Contingency Analysis Dialog is open, the oneline diagram's local menu will contain an additional option labeled *Set As Contingency Reference*. When you choose this option, you establish the case in memory at that time as the reference or starting point for future contingency analysis runs. You might want to do this, for example, if you are searching for ways to mitigate a particular contingency and find an operating guide that is successful. You may then model that operating guide as being in place when you model the rest of the contingencies in the list.

Chapter 13

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 (page 229) for details.

- Save all oneline diagrams and strip charts you wish to publish as jpeg files. See Saving Images As Jpegs (page 231) 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.

The rest of this discussion will explain how to accomplish these tasks.

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.

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.

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.

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.

Chapter 14

Script Files

Script files are used to automatically control the Simulator.

This chapter covers the following:

- Overview of script files, including how they are created, played and cleared

- Script file format

Script Files

One of the important capabilities of Simulator is the ability to schedule particular events to occur at specific times during the simulation. This is accomplished using script files. As the name implies, a script file contains specific actions which take place automatically at specific times during the simulation. Using script files, just about any action a user can do manually can be set up to occur automatically. In addition, it is possible to add written text annotations which appear on the screen at various times throughout a simulation. Thus an entire demonstration of a simulation case can be set up beforehand and then simply played for an audience. Note that while a script file is playing, the user still has the complete ability to interact with the simulation.

Identifying Script Files

The script file for a case is denoted with the *.SCP file extension. The script file should always be in the same directory as the *.PWB and *.PWD files. Note, however, that a case need not have a script file.

Playing a Script File

Whenever you load a case into Simulator, the script is loaded from the *.SCP file (if a script file exists). The script is automatically played when the simulation is started. You can tell that a case has an associated script in two different ways. First, the expression '(using script)' is appended to the simulation title. Second, in the Case Summary dialog box the line 'Case uses a Script' appears on the line immediately before the OK button. While a script is playing, you still have complete ability to interact with the simulation.

You can also manually identify a script file to play by selecting **Simulation, Open Script File** from the main menu in Run Mode.

Clearing the Script

At any time you can clear the script by selecting **Simulation, Clear Script** from the main menu. Clearing the script removes all future scripted events for the current case, but does not delete the *.SCP file.

Creating a Script File

The script file itself is an ASCII file and can therefore be modified using any type of text editor program. See Script File Format (page 238) for more details.

Script File Format

This section describes the actual format used with the Script Files.

Note, we anticipate changing this format in future releases of Simulator. However, if the format does change, we will provide easy conversion between the new format and the format described here.

Script files can be created either manually using a text editor, or automatically using the record feature of Simulator. The input file format for script files is free format (with rare exception) with the fields occurring in the following order:

Agent ID: This integer field is intended to identify the requesting agent. Simulator currently ignores the value of this field.

Time: Simulation time at which the input event should be executed. The format of this field is hh:mm:ss where hh is simulation hours, mm is simulation minutes and ss is simulation seconds. The seconds field can include fractional seconds or it can be excluded. Events are executed as soon as possible at or after the simulation has reached the specified time. Therefore to execute an event as soon as possible specify the time as less than the current simulation time (such as 00:00:00).

Event Class: The event class is the general category of the event, being specified as a case independent character string. Since the fields following the event class are event class dependent, they are described under the corresponding event class type. The following event classes are valid:

AREA   Used to specify events that affect a particular operating area. The next field is the area number (from 1 to 100). Following the area number is the case independent action field. Valid actions are:

    AGC   The next integer field specifies the area AGC status: 1 - used to place the area on automatic generation control, 0 - used to take the area off automatic generation control.

BRANCH   Used to specify events that affect a particular transmission line or transformer (i.e., a branch). The next three fields specify the "from bus" number, the "to bus" number and the circuit identifier (from 1 to 99). Following the circuit identifier is the case independent action field. Valid actions are:

    CLOSE   Closes the branch's circuit breakers, energizing the device (providing at least one of the terminal buses is already energized).

    IMPEDANCE   Sets the resistance and reactance of the branch (all values are in per unit).

    OPEN   Opens the branch's circuit breakers, de-energizing the device.

    RATING   Sets the first (or only) MVA rating for the branch.

    SUSCEPTANCE   Specifies the total charging susceptance (in per unit).

    TAPRATIO   Specifies a new value for the off-nominal turns ratio. Following IEEE common format convention, taps are assumed to be at the "from bus." The next field specifies the tap position.

    TAPANGLE   Specifies a new value for the phase shift angle (in degrees). Following IEEE common format convention, taps are assumed to be at the "from bus." The next field specifies the tap position.

CAP	Used to specify events that affect a capacitor at a particular bus. The next field is the bus number (from 1 to 99999) to which the capacitor is connected. Following the bus number is the case independent action field. Valid actions are:

CLOSE	Close the device's circuit breaker, energizing the device (provided the bus is already energized).
MVRBASE	Used to specify a new value for the capacitor's nominal Mvar injection (assuming 1.0 per unit terminal voltage). The next field specifies the new value of the capacitance (in Mvar).
OPEN	Open the device's circuit breaker de-energizing the device.

GEN	Used to specify events that affect a generator at a particular bus. The next field is the bus number (from 1 to 99999) to which the generator is connected. Immediately following the bus number the next field is a single character machine identifier (0 through 9 or A through Z). Next is the case independent action field. Valid actions are:

AGC	The next integer field specifies the generator's automatic generation control (AGC) status: 1 - used to place the generator on AGC, 0 - used to take the generator off of AGC.
AVR	The next integer field specifies the generator's automatic voltage regulation (AVR) status: 1 - used to place the generator on AVR control, 0 - used to take the generator off of AVR control.
CLOSE	Close the device's circuit breaker, energizing the device (provided the bus is already energized).
COSTMODEL1	Simulator uses the following cost formula (for generator i)

$$C_i(P_{gi}) = a_i + b_i P_{gi} + c_i (P_{gi})^2 + d_i (P_{gi})^3 \qquad \text{\$/hr}$$

This field specifies the parameters a_i, b_i, c_i, and d_i.

MVR	Specifies the reactive power output of the generator in Mvar. This value can only be specified when the generator is off of AVR control, and is restricted to be between the minimum and maximum reactive power limits
MVRDELTA	Change the generator's reactive power output by the amount specified in the next field (in Mvar). This action only has an affect when the generator is off of AVR control.
MVRLIMITS	Specifies new minimum and maximum reactive power limits (in Mvar).
MW	Specifies the real power output of the generator in MW. This value is restricted to be between the minimum and maximum real power limits.
MWDELTA	Change the generator's real power output by the amount specified in the next field (in MW). This action always takes the generator off of Area AGC control.
MWLIMITS	Specifies new minimum and maximum real power limits (in MW).
OPEN	Opens the device's circuit breaker de-energizing the device.

LOAD	Used to specify events that affect the load at a particular bus. The next field is the bus number (from 1 to 99999) to which the load is connected. Following the bus number is the case independent action field. Valid actions are:

CLOSE	Close the device's circuit breaker, energizing the device (provided the bus is already energized).
MVR	Specifies a new reactive power load. The base reactive power load is then set equal to this new value divided by the load multiplier for the area.

MVRBASE	Specifies a new base reactive power load. The actual reactive power load is then equal to the base value times the load multiplier for the area.
MW	Specifies a new real power load. The base real power load is then set equal to this new value divided by the load multiplier for the area.
MWBASE	Specifies a new base real power load. The actual reactive power load is then equal to the base value times the load multiplier for the area.
OPEN	Opens the device's circuit breaker de-energizing the device.

SIMULATION   Used to specify events that affect the simulation. Following the event class field is the case independent action field. Valid actions are:

ANIMATEDFLOWS	Specifies whether the flow of real power should be animated on the oneline display; 1 - show animated flows, 0 - do not show animated flows.
ANNOTATE	Writes some text at a particular location on the computer's screen for a particular amount of time. The next fields are the number of lines of text to display (always 1 in the current version), the amount of time in simulation minutes to display the text, the font size of the text, the location on the screen to display the text (given in percentage of the screen width and percentage of the screen height, measured from the upper left-hand corner), and in single quotes the text to display.
BEEP	Causes the computer's speaker to beep once.
CONTINUE	Continues a paused simulation. The continue command is always immediately executed (since simulation time does not advance in a paused simulation).
ENFORCEGENRAMPLIMITS	Specifies whether generator ramp MW ramp rate limits should be enforced; 1 - enforce limits, 0 - do not enforce limits.
ONELINEDETAIL	Specifies the amount of information shown on the oneline display; 0 corresponds to "Minimal", 1 corresponds to "Moderate", while 2 corresponds to "Complete."
PAUSE	Pauses the simulation for the specified number of seconds. For an indefinite pause, specify the number of seconds as a negative number. An indefinite pause can be continued by selecting **Continue** from the **Simulation** menu.
RESET	Resets the simulation to the starting time and restarts the simulation. This will allow continuous play of a script file.
SPEED	Determines the speed of the simulation relative to real-time.
UNEXPECTEDEVENTS	Specifies the frequency of unexpected events; 0 corresponds to "Never", 1 corresponds to "Occasionally", while 5 corresponds to "Often."
VOLTPU	Displays the voltages on the oneline in per unit.
VOLTKV	Displays the voltages on the oneline using actual kV.

TRANSACTION   Used to specify real power transactions between control areas. The next two fields specify the buying and selling areas respectively. Following is the case independent action field. Valid actions are:

CANCEL	Cancels a transaction between the two areas. In order to uniquely identify the transaction to cancel, the next fields must specify the original ending time for the transaction (using the same format as the time field), the MW amount of the transaction, and the price (in $/MWH). If a transaction with those parameters does not exist, this command is ignored.

MWFIRM		Sets up a firm transaction between the two areas. The next fields specify the ending time for the transaction, the MW amount of the transaction, and the price (in $/MWH).
MWNONFIRM		Sets up a non-firm transaction between the two areas. The next fields specify the ending time for the transaction, the MW amount of the transaction, and the price (in $/MWH).

TRANSFORMER		The transformer class has the same format and actions as the branch class. Transformers also have the following additional actions:
	AVR	The next integer field specifies the LTC transformer's automatic voltage regulation (AVR) status: 1 - used to place the transformer on AVR control, 0 - used to take the transformer off AVR control.
	AWR	The next integer field specifies the phase shifting transformer's automatic watt regulation (AWR) control status (so that the MW flow through the phase shifter is controlled):1 - used to place the transformer on AWR control, 0 - used to take the transformer off AWR control.
	MWLIMITS	Specifies the minimum and maximum limits for the real power flow through the phase shifter (in MW) at the regulated bus.
	PHASESTEP	Phase shifter transformer step increment (in degrees).
	PHASELIMITS	Specifies the minimum and maximum phase shift angles (in degrees).
	RATIOSTEP	LTC transformer turns ratio step increment.
	RATIOLIMITS	Specifies the minimum and maximum off-nominal tap ratios for LTC transformers.
	REGBUS	Specifies the number of the regulated bus for either an LTC transformer or a phase shifting transformer. For a phase shifter the regulated bus must be one of the transformer terminal buses.
	VOLTLIMITS	Specifies the minimum and maximum voltage limits at the regulated bus for an LTC transformer.

WINDOW		Used to specify events that affect the windows shown on the screen. The next field is the case independent window name. In PowerWorld Simulator the following are valid window names:
	ACE	Ace chart
	AREA	Area Transactions/Information Display
	AREAINFO	Area Information Dialog
	AVGMWHCOST	Average Cost per MWH chart
	LOADGEN	Load and Generation chart
	ONELINE	Oneline diagram window
	SCHEDTRANS	Scheduled Transactions chart
	SYSFREQ	System Frequency chart

Following the window name is the ID field for the window. For all windows, except **AREA, ONELINE,** and **SYSFREQ**, the ID is the area number; for **ONELINE** the ID field should be the name of the display file (e.g., B2.PWD), for **AREA** and **SYSFREQ** the ID should be zero (its value is currently ignored).

The size of the windows and their position on the screen is always specified as a percentage of the screen size. This permits their use on displays with different resolutions. Following the ID field is the case independent action field.

CLOSE	Closes the window.
MAXIMIZE	Maximizes the window so it fills the entire screen.
MINIMIZE	Reduces the window to an icon.
MOVE	Moves the window. Specify the percentage left and top of the upper left-hand corner of the window.
OPEN	Opens a new window. Specify the percentage left and top of the upper left-hand corner of the window, and the percentage width and height of the window.
PAN	Pans to a new screen location. Specify the *x* and *y* coordinates of the new screen center. The last field EndTime is optional. If EndTime is greater than Time, the screen is linearly panned to the specified location starting at Time and ending at EndTime. If EndTime is less than or equal to Time, or omitted, the screen is immediately panned to the new location.
RESIZE	Re-sizes the window. Specify the new percentage width and height of the window.
RESTORE	Restores a window to its former size after either a maximize or a minimize.
VIEW	Pans and zooms to the new oneline "VIEW" location. The next field specifies the name of the oneline view enclosed in quotes. The optional last field, EndTime, is the ending time value. If EndTime is greater than Time, the screen is linearly panned to the specified location starting at Time and ending at EndTime. If EndTime is less than or equal to Time, or omitted, the screen is immediately panned to the new location.

For graph/stripchart windows the following additional actions are also valid:

XAXISSCALE	Changes the scaling used on the x-axis of graph/stripchart windows. For graphical windows specify the new minimum x value, maximum x value, and the number of intervals shown on the x-axis. For strip chart windows (such as ACE, AVGMWHCOST, LOADGEN, SCHEDTRANS, and SYSFREQ) specify the number of minutes shown on the x-axis, another number (which is ignored), and the number of intervals shown on the x-axis.
YAXISSCALE	Changes the scaling used on the y-axis of graph/stripchart windows. Specify the new minimum y value, maximum y value, and the number of intervals shown on the y-axis.

Examples: **WINDOW ACE 2 OPEN 50 50 40 30**

creates an ACE window showing information for area number 2, where the upper left-hand corner of the window is in the middle of the screen, with a width equal to 40% of the screen size and a height equal to 30% of the screen size.

WINDOW ACE 2 YAXISSCALE -20 20 4

changes the scaling of the y-axis for the ACE window showing information for area number 2, with a new minimum of -20 MW, a maximum of 20 MW, and 4 intervals (-20, -10, 0, 10 and 20 MW).

WRAREA Used to specify events that write information about an operating area to the output file. The next field is the area number (from 1 to 100). Following the area number is the case independent action field.

ACE	Writes the area's ACE in MW.
FLOW	Writes the net MW and Mvar flow (interchange) out of the area.
GEN	Writes the net MW and Mvar generation for the area.

LOAD	Writes the net MW and Mvar load for the area.
LOSSES	Writes the net MW and Mvar losses for the area.

WRBRANCH Used to specify events that write information about a particular transmission line or transformer (i.e., a branch). The next three fields specify the "near bus" number, the "far bus" number, and the circuit identifier (from 1 to 99). All flow values are assumed to be at the "near bus" end of the line. Following the circuit identifier is the case independent action field. Valid actions are:

AMP	Writes the current magnitude in amps at the near bus end of the branch.
IMPEDANCE	Writes the per unit resistance and reactance of the branch.
MVR	Writes the reactive power flow in Mvar out of the branch at the near bus end.
MW	Writes the real power flow in MW out of the branch at the near bus end.
RATING	Writes the first (or only) rating of the line in MVA.
STATUS	Writes the status of the line circuit breakers: 1 - breakers are closed, 0 - breakers are open.
SUSCEPTANCE	Writes the total per unit charging susceptance of the branch.
TAPRATIO	Writes the off-nominal turns ratio. Following IEEE common format convention, taps are assumed to be at the "from bus." The next field specifies the tap position.
TAPANGLE	Writes the phase shift angle (specified in degrees). Following IEEE common format convention, taps are assumed to be at the "from bus." The next field specifies the tap position.

WRBUS Used to specify events that write information about a bus to the output file. The next field is the bus number. Following the bus number is the case independent action field.

STATUS	Writes the status for the bus: 1 - bus is energized, 0 - bus is not energized (i.e., dead).
VOLTKV	Writes the bus voltage magnitude in kV.
VOLTPU	Writes the per unit bus voltage magnitude.
VOLTRAD	Writes the bus voltage angle (in radians).

WRCAP Used to specify events that write information about a capacitor to the output file. The next field is the bus number. Following the bus number is the case independent action field.

MVR	Writes the actual reactive power output of the capacitor in Mvar.
MVRBASE	Writes the base (nominal) Mvar value for the capacitor (i.e., reactive power output assuming 1.0 per unit terminal voltage).
STATUS	Writes the status for the capacitor's circuit breaker: 1 - breaker is closed, 0 - breaker is open.

WRCASE Used to specify events that write information about the entire case.

IEEE	Writes the case power flow data into the specified file (you must give the complete pathname) using the IEEE common format.
PFLIST	Writes a listing of all the power flow into all the buses in the system. The format of this file is similar to the file used in the PowerWorld System Power Flow List.

	PTI	Writes the case power flow data into the specified file (you must give the complete pathname) using the PTI PSS/E format.

WRGEN   Used to specify events that write information about a bus to the output file. The next field is the bus number, followed by the single character generator ID. Then comes the case independent action field.

	AGC	Writes the AGC status.
	AVR	Writes the AVR status.
	COSTMODEL1	Simulator uses the following cost formula (for generator i)

$$C_i(P_{gi}) = a_i + b_i P_{gi} + c_i (P_{gi})^2 + d_i (P_{gi})^3 \qquad \$/hr$$

This field writes the parameters a_i, b_i, c_i, and d_i.

	MVR	Writes the reactive power output in Mvar.
	MVRLIMITS	Writes the minimum and maximum reactive power limits (in Mvar).
	MW	Writes the real power output in MW.
	MWLIMITS	Writes the minimum and maximum real power limits (in MW).
	STATUS	Writes the status for the generator's circuit breaker: 1 - breaker is closed, 0 - breaker is open.

WRLOAD   Used to specify events that write information about a load to the output file. The next field is the bus number. Following the bus number is the case independent action field.

	MVR	Writes the reactive power load in Mvar.
	MVRBASE	Writes the base reactive power load in Mvar. The actual reactive power load is then equal to the base value times the load multiplier for the area.
	MW	Writes the real power load in MW.
	MWBASE	Writes the base real power load in MW. The actual reactive power load is then equal to the base value times the load multiplier for the area.
	STATUS	Writes the status for the load's circuit breaker: 1 - breaker is closed, 0 - breaker is open.

ZOOM   Zooms to a new screen location. Specify the new zooming percentage level (nominal value is 100). The optional last field, EndTime, is the ending time value. If EndTime is greater than Time, the screen is linearly panned to the specified location starting at Time and ending at EndTime. If EndTime is less than or equal to Time, or omitted, the screen is immediately panned to the new location.

Chapter 15

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 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 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 a 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.

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.

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.

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 Explorer's 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.

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 on-line use, although they can be used in text format as reproduced here.

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 (page 249). Or if you're interested in trial-and-error learning, you may just wish to open one of the sample cases and start learning.

Tutorial - Creating a New Case

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 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 (page 257).

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 oneline diagram. Oneline diagrams are used in power system analysis to represent the actual three-phase power system using a single line to represent each three-phase device. The yellow background represents the blank page on which you will be drawing a new oneline diagram.

Tutorial : Inserting a Bus

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices are connected together. In building a power system model using Simulator, you will draw buses onto the oneline diagram, attach devices such as generators and loads to the buses, and connect different buses together with transmission lines and transformers.

Inserting buses on the oneline 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 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 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 oneline 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.

Tutorial: Inserting a Generator

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.
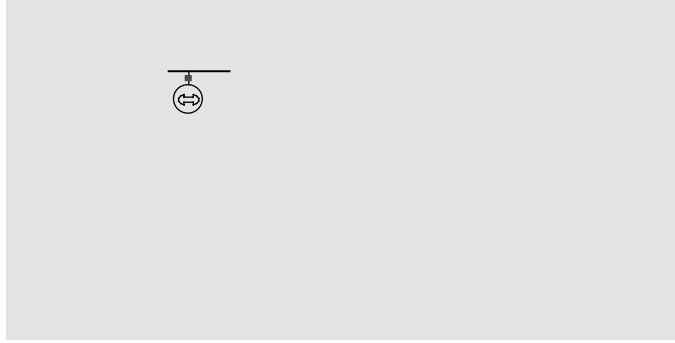
Tutorial: Saving the Case

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.

Tutorial: Entering a Second Bus with Load

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.
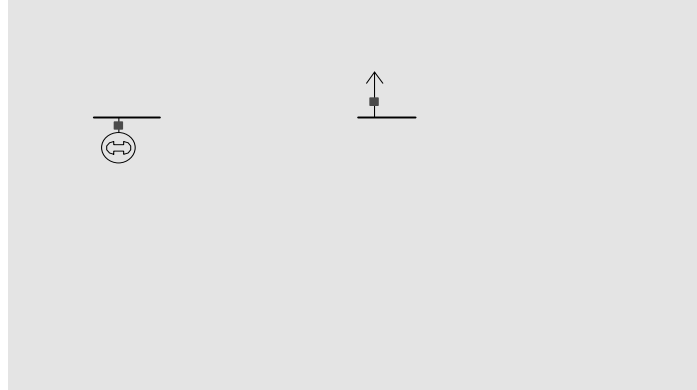
Tutorial: Inserting a Transmission Line

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.
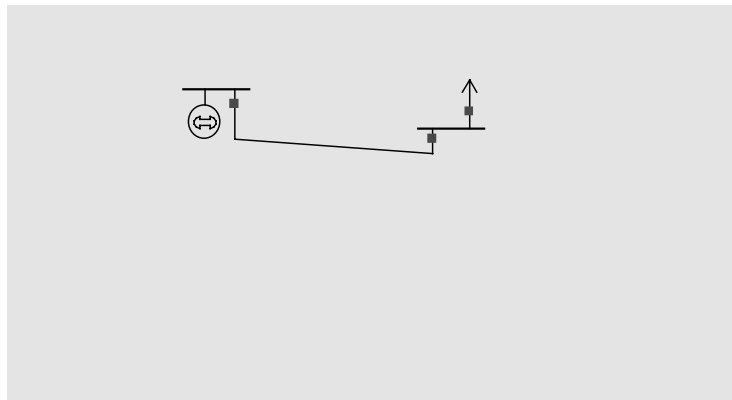
Tutorial: Solving the Case

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.

Tutorial: Adding a New Area

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.
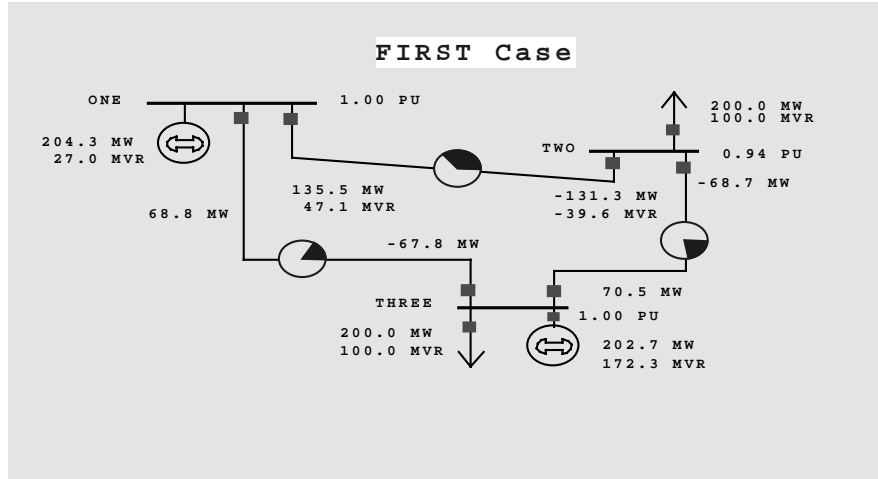
Tutorial: Case Customization

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 *Oneline Display Options Dialog* by selecting **Options/Tools, Oneline 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 Oneline 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.

Tutorial - Starting with an Existing Case

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 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 (page 249).

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.

Tutorial: Loading an Existing Power Flow File

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.

Tutorial: Case Information Displays

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 Display Filters information display is another important article. The Display Filters feature restricts the contents of other case information displays to certain areas or zones. This is particularly useful for large cases. Open the Display Filters display by selecting **Case Information, Display Filters** from the main menu, or use the **Display 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 display filter status, simply left-click on the area's display 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 display 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 display 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.

Tutorial: Solving the Case

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.

Tutorial: Building Onelines

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.

Tutorial: Entering a Bus

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.

Tutorial: Automatic Line Insertion

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.

Tutorial: Panning and Zooming

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.

Tutorial: Adding Background

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.

Tutorial: Simulating the Case

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, Oneline 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.

Tutorial: Run-time Object Dialogs

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.

Tutorial: Oneline Local Menu

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.

Tutorial: Area

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. If you wish to view the limit violations for a particular area, you can set the area's **Report Limits** field to *Yes* to enable reporting of the limit violations for the area.

Tutorial: Limit Violations

Once you have defined the areas for which to report limit violations, 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. The displayed violations will correspond only to areas whose **Report Limits** field is set to *Yes* on the Area Records display.

In the Bus Voltage Violations report, the entries are sorted by per-unit voltage by default. 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.

In the Line/Transformer Violations report, the entries are sorted by percentage overload, by default. Most of the options that were available with the Bus Voltage Violations report area available with the Line/Transformer Violations report, too. All line and transformer limits are entered in MVA. However, if you wish to view the limits in equivalent amps, select the option to treat the transmission lines limits as equivalent amps from the top of the display.

Interface records are used to monitor flow on a group of lines or transformers. Interfaces are often used as proxies for other power system limits, such as transient stability or voltage stability limits. Interfaces are not defined in most other power flow data formats, but Simulator provides an easy way to add them to the case. To define an Interface Record, select **Case Information, Interfaces** from the main menu to view the **Monitored Interface** display. Right-click on the display and select **Insert** from the local menu. This displays the **Monitored Interface Dialog**, which is used both to display information about existing interfaces and to define new interfaces. For the MAIN example, we will set up an interface showing the flow on two parallel transformers. Name the interface FCHXFS and set its limit as 80MW. Change the **From Bus Number** field to 39821 and the **To Bus Number** field to 39831. Set the **ID** field to 1. Click on **New Element** to add a new element to the interface. The interface element should now be shown in the **Interface Elements Description** field. Repeat this procedure to add the other transformer, except change the **ID** field to 2. The dialog will now consist of the two transformers. Select *OK* to add the Interface and close the dialog.

Tutorial: Other Case Information Displays

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