NETWARS Network Model Configuration and Navigation

2005-1 Final User's Guide (OPNET 229)

Contract DASW01 03 D 0008

June 27, 2005

Prepared for:	Prepared by:
Defense Contracting Command - Washington	OPNET Technologies, Inc.
Washington, DC 50310-5200	7255 Woodmont Avenue
	Bethesda, MD 20814-7904

OPNET®
Optimum Network Performance

Identification

Document Identification

> Document Title: NETWARS Network Model Configuration and Navigation
> Version: Final (OPNET 229)

Software Identification

> Product Name: NETWARS
> Product Release: 5.1

Documentation Conventions

> This documentation uses specific formatting and typographic conventions to present the following types of information:
>
> • Objects, examples, and system I/O
>
> • Object hierarchies
>
> • Computer commands
>
> • Lists and procedures

Objects, Examples, and System I/O

> • Directory paths and file names are in standard Courier typeface:
>
> ```
> C:\Netwars\User_Data\Projects
> ```
>
> • Function names in body text are in italics:
>
> *op_dist_outcome()*
>
> • The names of functions of interest in example code are in bolded Courier typeface:
>
> ```
> /* determine the object ID of packet's creation module */
> src_mod_objid = op_pk_creation_mod_get (pkptr);
> ```
>
> • Variables are enclosed in angle brackets (< >):
>
> ```
> <NETWARS path>\Scenario_Builder\op_admin\err_log
> ```

Object Hierarchies

> Menu hierarchies are indicated by right angle brackets (>); for example:
>
> Edit > Preferences > Advanced

Computer Commands

These conventions apply to Windows systems and navigation methods that use the standard graphical-user-interface (GUI) terminology such as click, drag, and dialog box.

• Key combinations appear in the form "press **<button>+x**"; this means press the **<button>** and **x** keys *at the same time* to do the operation.

• The mouse operations *left-click* (or *click*) and *right-click* indicate that you should press the left mouse button or right mouse button, respectively.

Lists and Procedures

Information is often itemized in bulleted (unordered) or numbered (ordered) lists:

• In bulleted lists, the sequence of items is not important.

• In numbered lists, the sequence of items is important.

Procedures are contained within procedure headings and footings that indicate the start and end of the procedure. Each step of a procedure is numbered to indicate the sequence in which you should do the steps.

Document Revision History

Release Date	Product Version	Chapter	Description of Change
June 27, 2005	5.2 Final	2	Inserted new figure: Figure 2-4 Adding Units via Object Palettes
		3	- Added section "Using Configuration Utilities" to describe device configuration wizard for promina and multiplexer circuits.
			- Replaced term "subnet" with "OPFACS and Organizations" (where more appropriate to NETWARS users.)
			- In Procedure 3-4, clarified where to find specific protocol user's guides ~ provided as individual chapters in the OPNET Standard Models User Guide (NETWARS menu accessible.)
April 4, 2005	5.1 Draft	All	Preliminary version.

Contents

List of Figures

List of Procedures

1 Introduction

NETWARS Overview

The Command, Control, Communications, and Computer Systems Directorate of the Joint Staff, in partnership with the Defense Information Systems Agency, Directorate for Technical Integration Services, developed Network Warfare Simulation (NETWARS). NETWARS provides modeling and simulation (M & S) capabilities for measuring and assessing information flow through strategic, operational, and tactical military communications networks. Analyzing the results from NETWARS can provide considerable utility in determining which communication systems might be overloaded during selected times in a particular scenario, and can assist with making prudent acquisition planning decisions.

Document Overview

NETWARS provides a variety of features that are specifically designed to increase productivity and efficiency in your modeling and simulation projects. These features are designed to make it easier and faster for you to configure and navigate your network model, understand its configuration, and extract information to include in your reports.

This user's guide, *NETWARS Network Model Configuration and Navigation,* builds on information provided in the *NETWARS User's Manual* and *NETWARS Technical Reference Manual* to further describe and explain some of the powerful modeling and simulation features available to you in NETWARS.

This user's guide explores the following topics:

- File Management, including:
  - Managing file directories,
  - Using object libraries and palettes,
  - Accessing log files, product info and modules,
  - Managing and recovering project files, and
  - Refreshing and packaging.

- Configuration Helpers, including:
  - Editing attributes (for one object or a group of objects),
  - Configuring protocols, and
  - Using the Model Assistant to automate frequently applied scenario changes.

- Visualization and Navigation, including:

  — Finding and editing objects,

  — Viewing and editing demands,

  — Visualizing routing protocols and link loads,

  — Sizing and scaling icons,

  — Setting display preferences, and

  — Visualizing the effects of changing the network time.

- Reports, including:

  — Generating Summary Tables to view summary information for a scenario in table format,

  — Creating User-defined Report Tables to get a quick configuration overview of specific attributes of a number of network objects, and generate publishable HTML reports, and

  — Accessing Capacity Planning Web Reports to review the current state of a scenario after performing a capacity planning evaluation.

Referenced Documents

- *NETWARS User's Manual*, 2005-1 Final (OPNET 162), April 1, 2005.

- *NETWARS Technical Reference Manual*, 2005-1 Final (OPNET 176), April 1, 2005.

2   File Management

NETWARS network models are comprised of many significant files. The better you understand these files—where they are stored and what data they contain—the better you can manage your NETWARS environment.

Information provided in this chapter can help you to locate and identify important NETWARS files, as well as: manage the Organization and OPFAC Library, manage log files; display NETWARS product information, select product modules; backup NETWARS projects, and prepare a support log file to send to technical support if you should ever need assistance.

Manage File Directories

Two directories containing folders that, by default, store key NETWARS files include:

- `C:\Netwars\Scenario_Builder\` — Stores administrative and model files.

- `C:\Netwars\User_Data\` — Stores IER Text Files, OPFACS, Organizations, Projects, SB Data, Terrain Data, and Traffic Data files.

Administrative Files

Administrative files are located, by default, in the `c:\Netwars\Scenario_Builder\op_admin` folder. Files stored here include:

- Environment database (env_db) file — Stores NETWARS environment attributes (or *preferences*).

- Error log (err_log) and message log (session_log) files,

- 'bk' sub-directory containing backup files, and

- 'tmp' sub-directory containing temporary NETWARS-generated files.

Model Files

Model directories are located, by default, in the `c:\Netwars\Scenario_Builder\op_models` folder. You can change the default location where model files are stored by editing the 'mod_dirs' global environment attribute which contains the list of model directory pathnames.

Procedure 2-1   Setting Preferences for Model Directory Pathnames

1 In the System Editor or Scenario Builder, select **Edit > Preferences > Advanced**. The Advanced Preferences dialog box displays.

2   Locate the 'mod_dirs' attribute, and double-click it to open it in another window.
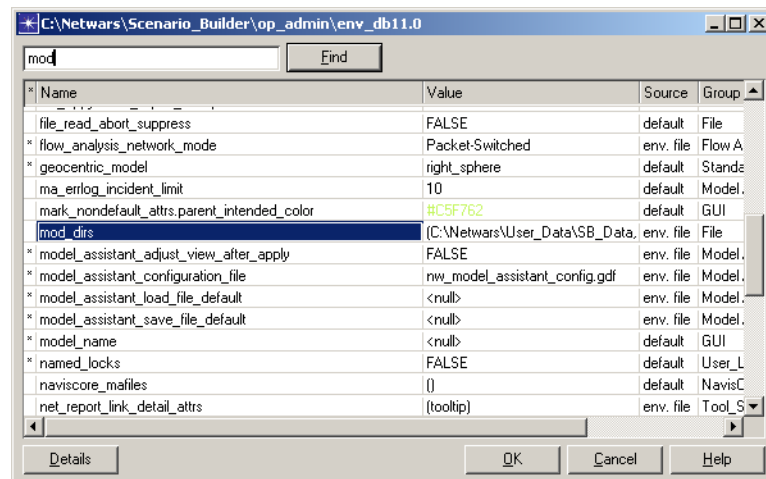
Figure 2-1   Advanced Preferences dialog box

3   Edit, add to, or delete from the list of model directory pathnames.
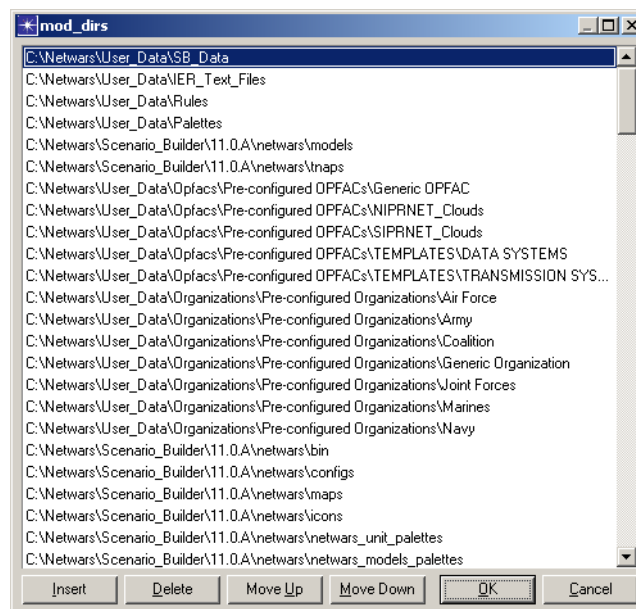
Figure 2-2   mod_dirs Attribute dialog box

4   Click **OK** to accept the attributes values and close the dialog box, or **Cancel** to close the dialog box without making any changes to the attribute.

End of Procedure 2-1

Project Files

Each project has a single `.prj` file in a project folder under `C:\Netwars\User_Data\Projects`. The project folder and the `.prj` file have the same name as the project itself. A NETWARS project may make use of other library files. Library files may be used by any number of projects. The only way to know which library files are included in a project is to examine the contents of the project.

Scenario Files

Each project consists of one or more scenarios. Each scenario is a subfolder under the main project folder. The scenario consists of a file with the extension '`.nt.m`' and possibly an XML file with the same base name.

Traffic Files

Demand Files

A demand file for each scenario, if one exists, would be located in the `NTWARS\data\User_Data\Projects\<Project>\<Scenario>` directory. Its name would be the Project Name followed by the Scenario Name connected with a plus ('+') and with an extension of '`.dm`'.

IER Text Files

All IER Text files are available for use by every scenario. IER Text files are located in the `C:\Netwars\User_Data\IER_Text_Files` directory and have an extension of '`.txt`'.

Organization Files

Organizations are represented by '`.nt.m`' files located in the `C:\Netwars\User_Data\Organizations` directory. The project may include any, all, or none of the template organizations.

OPFAC Files

The OPFACs used by a project/scenario may reside in any folder under the `C:\Netwars\User_Data\Opfacs` directory. NETWARS OPFACs consist of '`.nd.m`' and '`.nt.m`' files. The '`nt.m`' file includes all the OPFAC specific information while the '`.nd.m`' file is only necessary for an OPFAC to appear in a palette.

The names of the various files that relate to a specific OPFAC have similar base names, although not identical.

Manage the Organization and OPFAC Libary

In NETWARS, the Organization and OPFAC library is divided into two parts:

- Custom Library—Contains OPFACs and organizations that were created or modified by the user. All users have privileges to edit the contents of the custom library. You can make copies of the standard templates, store them in the custom library and then make changes to them. OPFACs and organizations in the custom library are called custom templates.

- Pre-Configured Library—Contains doctrinal OPFACs and organizations. OPFACs and organizations in the pre-configured library are called pre-configured templates. Only the local administrator has privileges to modify the contents of the pre-configured library, however any user can use its contents when building scenarios. A scenario can contain a combination of pre-configured and custom templates.

It is also possible to have sub-folders within the above set of folders. Use of sub-folders allows you to further categorize the organizations and OPFACs within a service folder.

The figure below shows an example of a library treeview with custom and pre-configured OPFAC and organization folders.
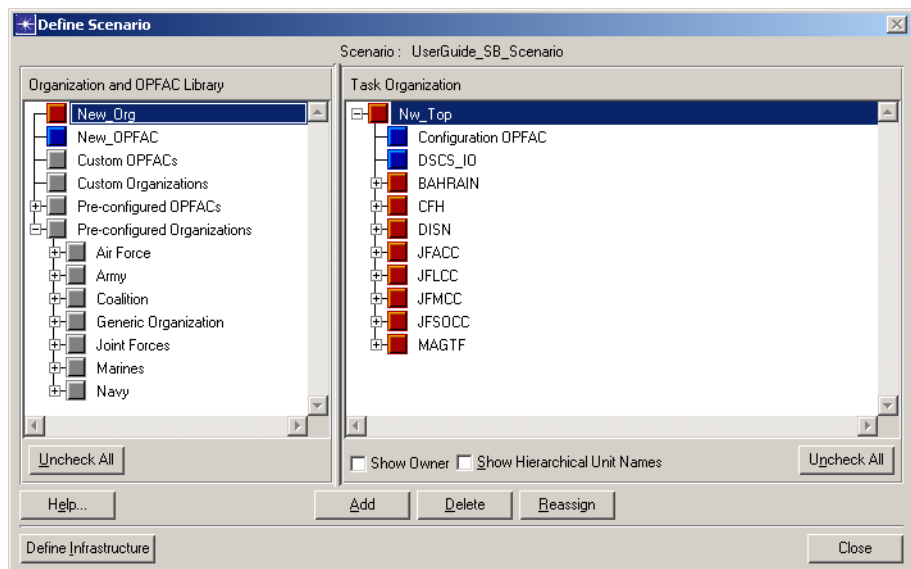
Figure 2-3   Library Treeview with Custom and Pre-Configured Folders

In addition to the above treeview interface, you can also use objects from the OPFAC and organization palettes to build the scenario. To build a scenario, you can drag-and-drop OPFACs and organizations from the palettes on to the workspace.
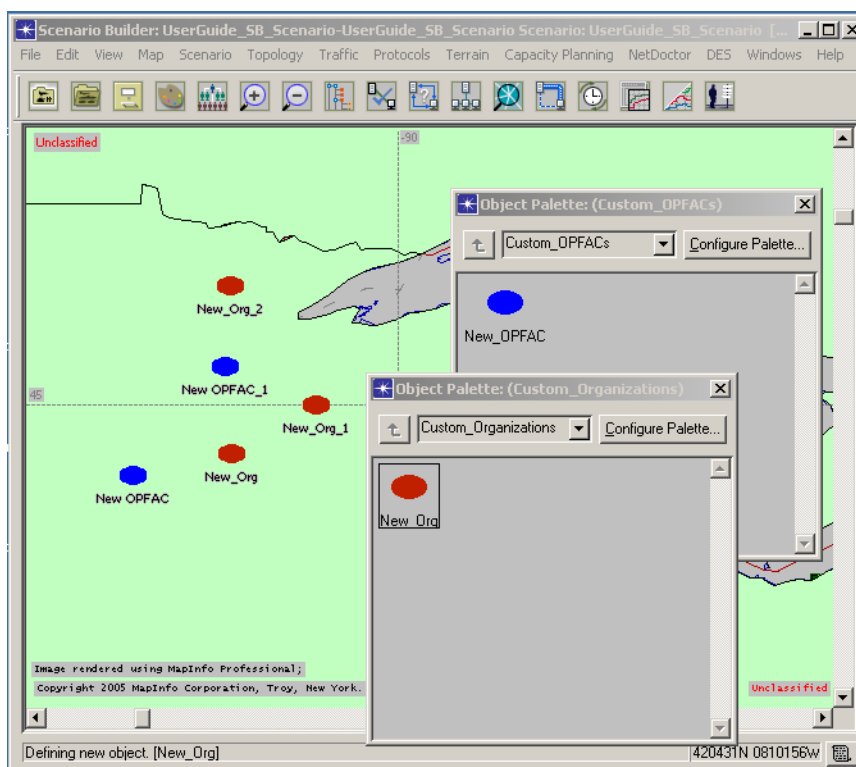
Figure 2-4   Adding Units via Object Palettes

Edit the Library

You can edit the contents of the Organization and OPFAC library. You can rename templates, save them under a different name or delete them from the library. Right-click menu options are provided for performing these operations.

If you login using the local administrator profile, you can perform the above operations on the pre-configured and custom sections of the library. If you login as a regular user, you can change the OPFACs or organizations in the custom section, but not the pre-configured section.

Procedure 2-2   Renaming a Template OPFAC/Organization

1 Right-click on a template OPFAC/organization in the library treeview and choose **Rename Template**.
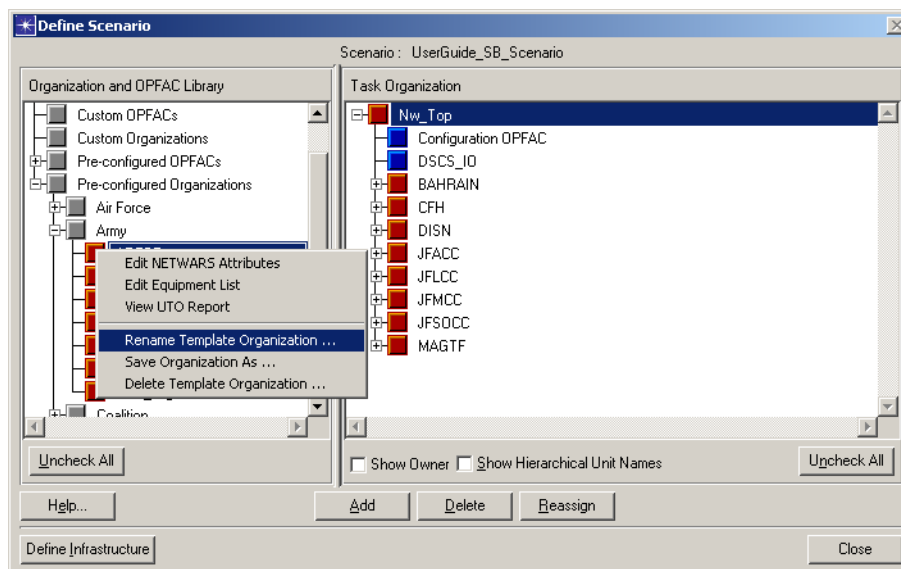
Figure 2-5   Renaming a Template OPFAC/Organization

2 Type a new name for the selected OPFAC/organization in the dialog box that displays, and change the folder to which the OPFAC/organization belongs. You can also create a new folder to store this OPFAC/organization.

The selected OPFAC/organization will be renamed, and if you changed the folder, the files will be moved to the new location on disk. The treeview is also automatically updated.

End of Procedure 2-2

Procedure 2-3   Saving a Template OPFAC/Organization As

1 Right-click on a template OPFAC/organization in the library treeview and choose **Save As**.

2   Type a new name for the selected OPFAC/organization in the dialog box that displays, and change the folder to which the OPFAC/organization belongs. You can also create a new folder to store this OPFAC/organization.

A copy of the selected OPFAC/organization will be made, and if you changed the folder, the files will be moved to the new location on disk. The treeview is also automatically updated.

End of Procedure 2-3

Procedure 2-4   Deleting a Template OPFAC/Organization

1   Right-click on a template OPFAC/organization in the library treeview and choose **Delete Template**.

2   Confirm the deletion as prompted to delete the selected OPFAC/organization from the library.

The selected OPFAC/organization will be removed from its current location on disk. The treeview is also automatically updated.

End of Procedure 2-4

Manage Error and Message Logs

The NETWARS error log contains descriptions and stack traces for recoverable and non-recoverable errors. This information is useful to NETWARS Technical Support in diagnosing and addressing application errors. The error log has a file name of 'err_log' and is stored, by default, in the `c:\Netwars\Scenario_Builder\op_admin` folder.

The NETWARS message log contains status updates that monitor the performance of NETWARS. The message log is also stored, by default, in the `c:\Netwars\Scenario_Builder\op_admin` folder.

Note—These log files accumulate messages over time. It is good practice to periodically check the sizes of the log files, and trim or empty the logs to prevent the files from becoming too large.

Procedure 2-5   Viewing the Error Log

1   In the System Editor or Scenario Builder, select **Help > Error Log > View**. The error log displays the most recent error log entries. If errors occur during the operation of NETWARS, they are logged in this error log file.
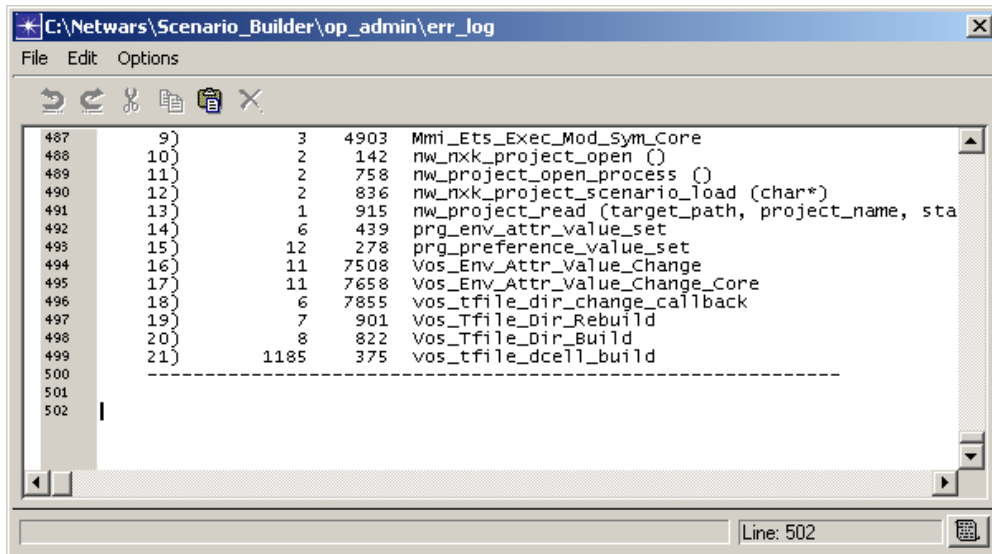
Figure 2-6   NETWARS Error Log

End of Procedure 2-5

Procedure 2-6   Clearing the Error Log

1   In the System Editor or Scenario Builder, select **Help > Error Log > Clear**. The Clear Error Log dialog box displays.
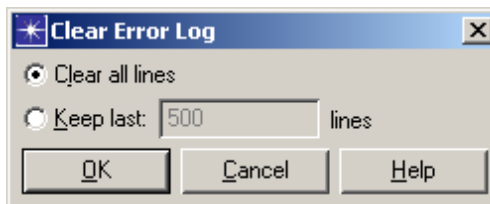
Figure 2-7   Clear Error Log dialog box

2   Click the **Clear all lines** radio button if you want to remove the entire contents of the log file, or click the **Keep last:** radio button if you want to type a number of most recent lines to keep (and remove earlier lines from the beginning of the log file so that only the specified number of lines remain.)

3 Click **OK** to clear the error log and close the dialog box, or **Cancel** to close the dialog box without making any changes to the error log.

End of Procedure 2-6

Procedure 2-7   Viewing the Message Log

1 In the System Editor or Scenario Builder, select **Help > Message Log > View**. The message log displays the most recent status messages.
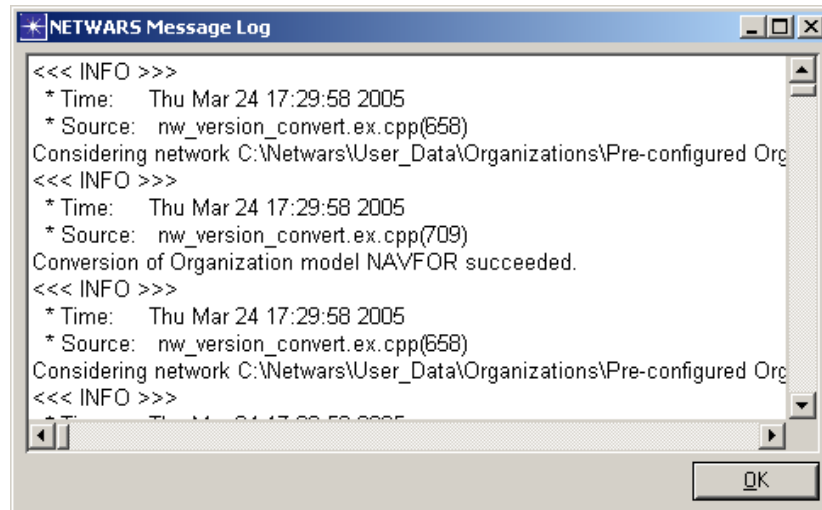
Figure 2-8   NETWARS Message Log

2 Click **OK** to close the NETWARS Message Log.

End of Procedure 2-7

Procedure 2-8   Clearing the Message Log

1 In the System Editor or Scenario Builder, select **Help > Message Log > Clear**. The message log clears immediately with no other dialog boxes or messages displayed.

End of Procedure 2-8

Display NETWARS Product Information

NETWARS provides important information about the OPNET core software version, the models version, and copyright information.

Procedure 2-9   Displaying NETWARS Product Information

1 In the System Editor or Scenario Builder, select **Help > About NETWARS**. The About NETWARS dialog box displays.

2 Click the About tab to display the information shown in the figure below:
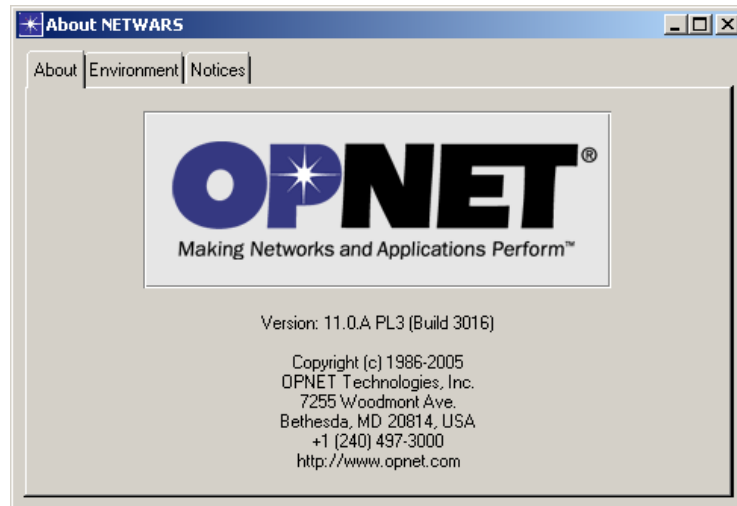
Figure 2-9   About NETWARS dialog box — About tab

3 Click the Environment tab to display the information shown in the figure below:
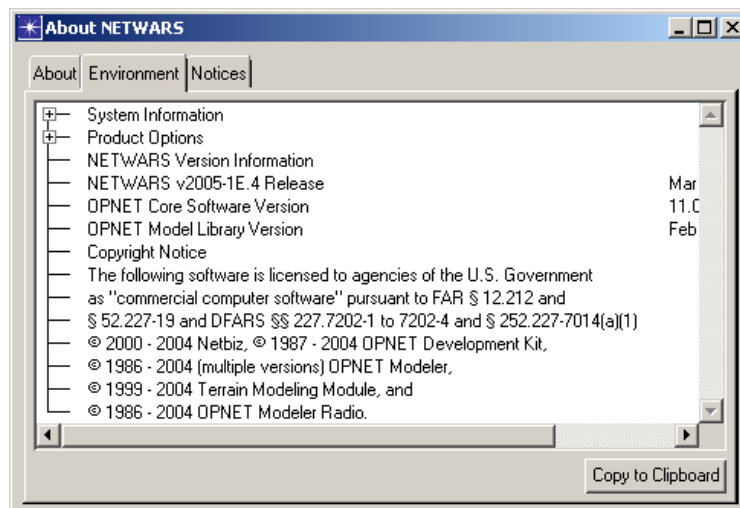
Figure 2-10   About NETWARS dialog box — Environment tab

4 Click the Notices tab to display the information shown in the figure below:
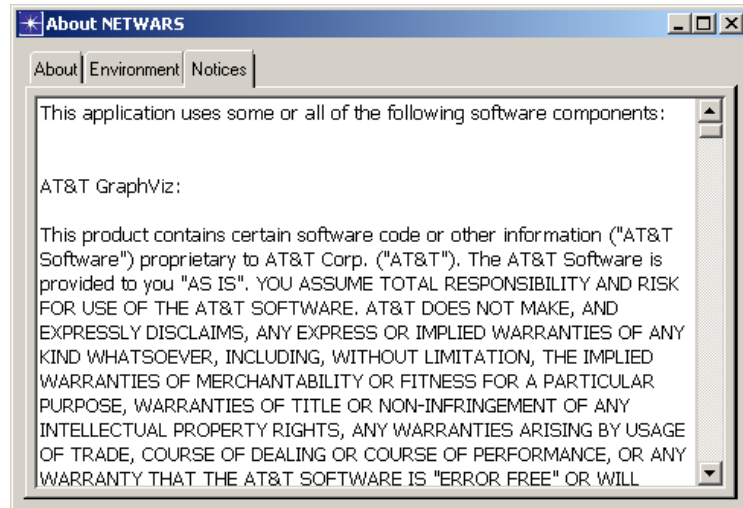
Figure 2-11   About NETWARS dialog box — Notices tab

End of Procedure 2-9

Select Product Modules

OPNET provides a variety of product modules that are specifically designed to increase productivity and efficiency in your modeling and simulation projects. Product modules that are licensed and available to you via NETWARS can be displayed and selected as discussed below.

Procedure 2-10   Selecting Product Modules

1 In the System Editor, select **License > Product Modules**. The Select Product Modules dialog box displays.
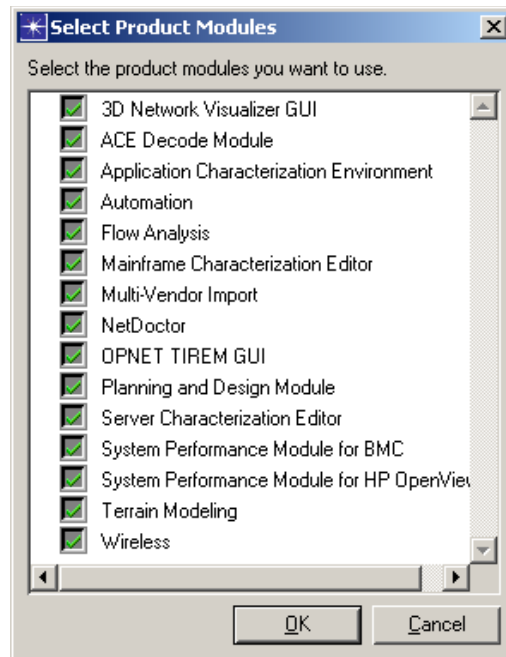
Figure 2-12   Select Product Modules dialog box

2 Click the checkboxes corresponding to the product modules you want to use.

3 Click **OK** to accept the checked modules and close the dialog box, or **Cancel** to close the dialog box without making any changes to your product module selection.

End of Procedure 2-10

Automatic Project File Backup

Project files are backed up automatically at a predetermined interval. You can modify the backup interval by editing the 'backup_interval' global environment attribute which contains the period (in minutes) of automatic file backups.

The backed up files have '`-backup<index>`' appended to the filename. You can load these backup files into Scenario Builder as needed.

Procedure 2-11   Setting Preferences for Backup Interval

1 In the System Editor or Scenario Builder, select **Edit > Preferences > Advanced**. The Advanced Preferences dialog box displays.

2 Locate the 'backup_interval' attribute, and click in the Value field.
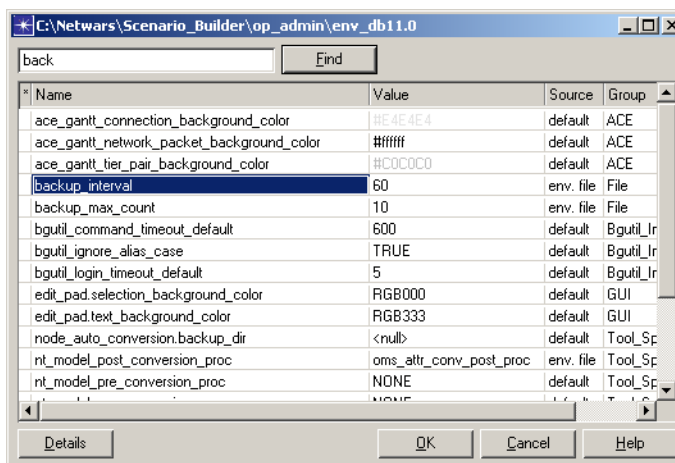
Figure 2-13   Advanced Preferences dialog box

3 Type a new value (in minutes) for the backup interval in the Value field.
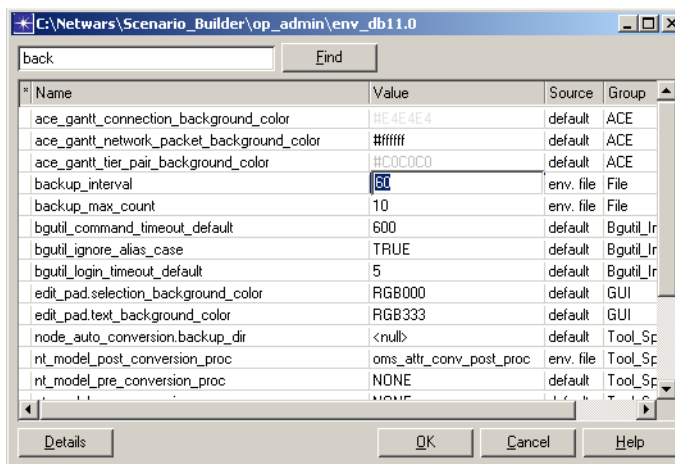
Figure 2-14   backup_interval Attribute Value field

4 Click **OK** to accept the attributes value and close the dialog box, or **Cancel** to close the dialog box without making any changes to the attribute.

End of Procedure 2-11

Generate Support Info

If you should ever need assistance, generate a support log file to send to NETWARS Technical Support. This can help Technical Support resolve your problem quickly and efficiently.

Procedure 2-12   Generating a Support Log File

1 In the System Editor, select **Help > Generate Support Info...**. The Save Support Info dialog box displays.
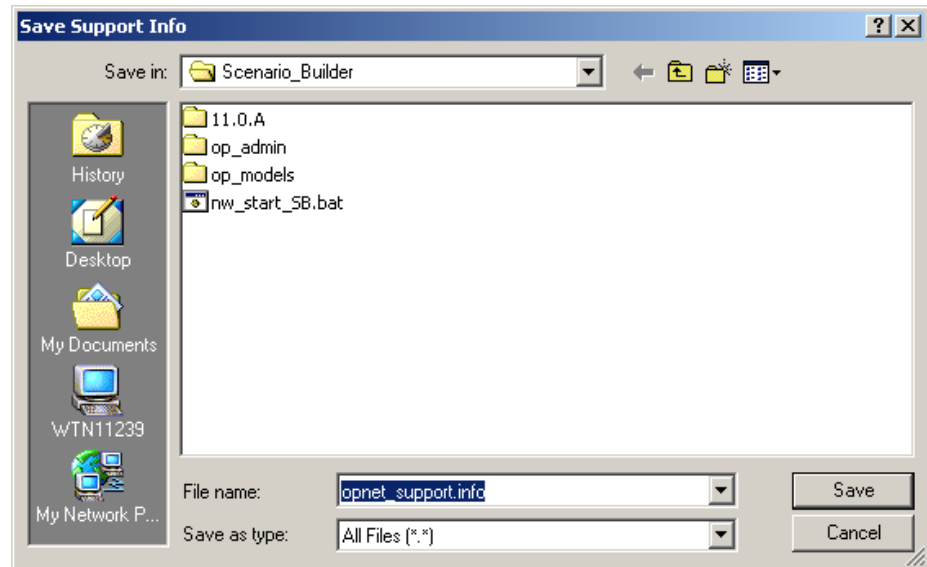
Figure 2-15   Save Support Info dialog box

2 Click **Save** to save a log file named opnet_support.info to the displayed directory.

3 When the log file has been created, the following message box displays. Click **OK** to close the message box.
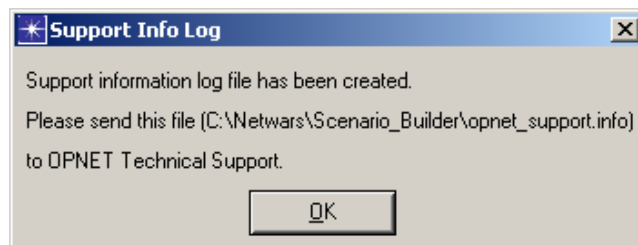
Figure 2-16   Support Info Log message box

End of Procedure 2-12

---

3   Configuration Helpers

NETWARS provides several features that make it easier and faster for you to configure your network model. Features discussed in this chapter can help you to edit attributes (for one object or a group of objects), configure protocols, and use the Model Assistant to automate frequently applied scenario changes.

Editing Object Attributes

Network objects have attributes that define their functionality, behavior, and physical characteristics. NETWARS provides several ways in which you can view, create, and edit network object attributes and their properties—for one object or for many objects—quickly and easily.

Editing Attributes for One Object

To edit the value of an attribute on a single object, use the following procedure.

Procedure 3-1   Edit Attributes for One Object

1   To view an object's attributes, right-click on the object to open the shortcut menu.

2   Select **Edit Attributes** to open the corresponding Attributes dialog box.
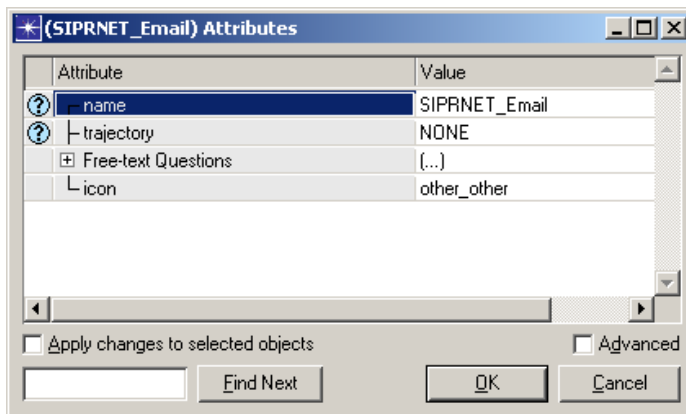
Figure 3-1   Attributes dialog box

The Attributes dialog box displays all *primary attributes* associated with the object. Primary attributes are those you use most often to define an object. For example, primary attributes specify the object's name and its underlying (node or link) model.

3   Set a new value for an attribute by typing a new value (or selecting one from the drop-down list if provided) in the corresponding Value field.

4   If the attribute that you want to edit does not appear in the main Attribute list, click the **Advanced** checkbox to display the object's *advanced attributes*.
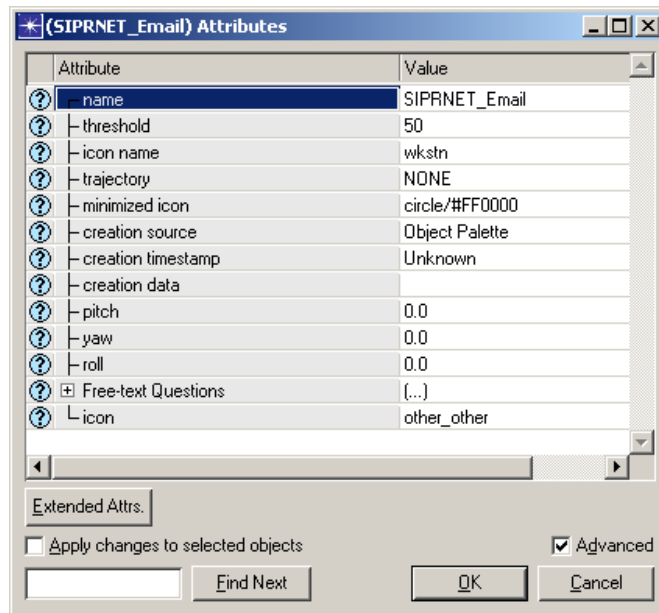
Figure 3-2   Advanced Attributes dialog box

Advanced attributes specify parts of the object definition that you might set less frequently. They typically define information such as an object's appearance and location in the workspace.

5   To find a specific attribute or value in the Attribute or Value lists, type a search term in the Find Next field and click **Find Next**. The first occurrence of the term will be highlighted.

6   To view details about an attribute, click the **?** button next to the attribute in the Attribute list. The Attribute Details dialog box displays.
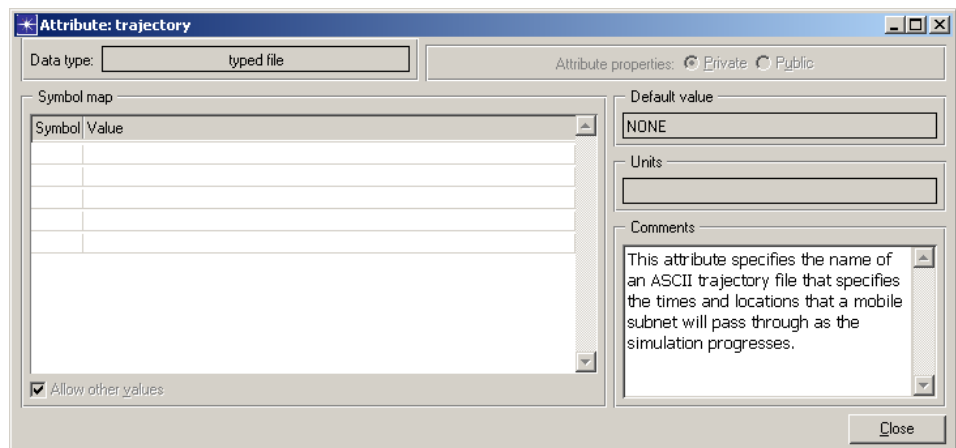
Figure 3-3   Attribute Details dialog box

6.1   Click **Close** to close the Attribute Details dialog box and return to the Attributes dialog box.

7 Click the **Apply changes to selected objects** checkbox if you want to apply the changes to an object's attribute values to both the object and all selected objects in the workspace.

8 To view extended attributes, click **Extended Attributes** in the Attributes dialog box. The Extended Attributes dialog box displays with a data table for creating new attributes.
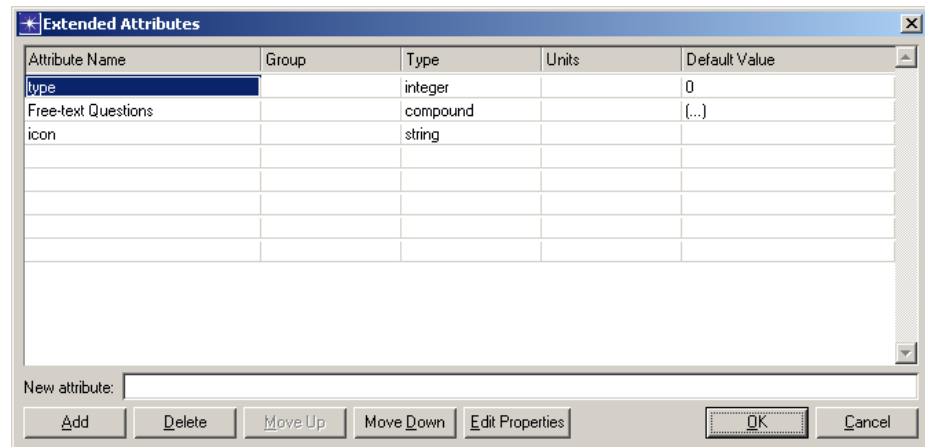
Figure 3-4   Extended Attributes dialog box

Extended attributes are optional model attributes that you can add to an object, thus further customizing its behavior in a network model.

8.1 Enter the name of the attribute in the New attribute field and click **Add**. You can also type directly in the data table by clicking in a cell in the Attribute Name column. The attribute is added to the table and automatically assigned a type and default value.

8.2 Under the Type column, select an appropriate data type from the pull-down menu.

8.3 In the Units column, type in the units to be associated with the attribute, if any.

8.4 In the Default Value column, type in a default value for the attribute.

9 To view, create, or edit the properties of an attribute, click **Edit Properties** in the Extended Attributes dialog box. The attribute's corresponding Properties dialog box displays.
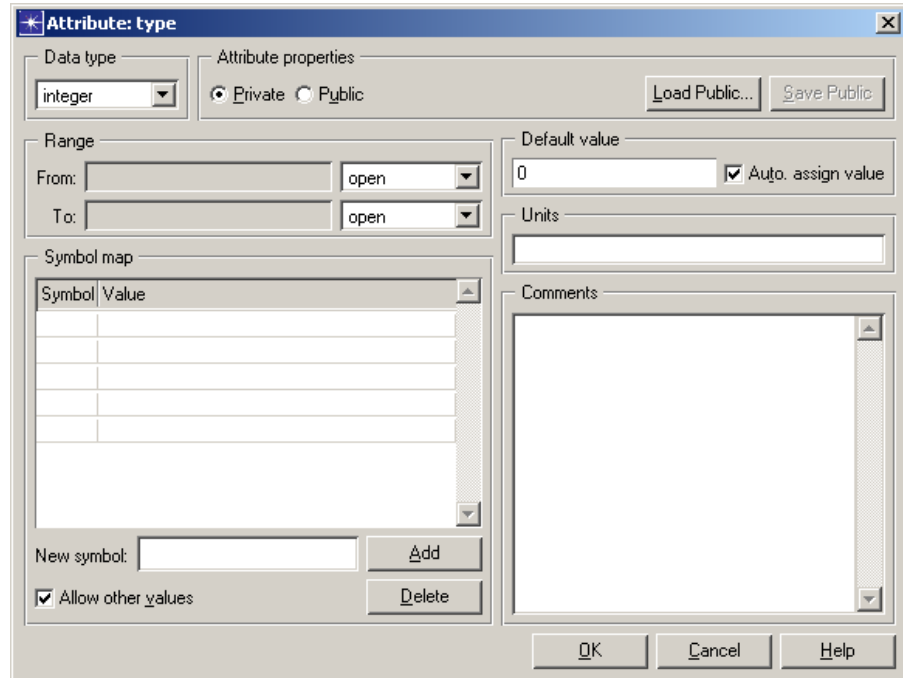
Figure 3-5   An Attribute's Properties dialog box

All attributes share a set of basic of properties that define the attribute, specifying information such as its name, data type, and allowed values. These properties are generally fixed, but can be given different values in some cases.

9.1 Set the type of the attribute in the Data type drop-down list.

9.2 Click the **Private** radio button if you want the property values to apply only to this attribute, or the **Public** radio button if you want the property values to be shared with other attributes. If the attributes are public, any changes to property values affect all attributes sharing these property values.

9.3 Click **Load Public** if you want to load a predefined set of public attribute properties.

9.4 Click **Save Public** if you want to save the current attribute properties to your default model directory with the suffix ".ad.m". This option is available only when you select the **Public** radio button.

9.5 Enter a range of allowable values for attributes of data type integer or double (this option does not apply to other data types) in the Range fields. For both the lower (From) and upper (To) limits, you can specify a numeric value and indicate that the value is included in the range ("inclusive") or not included ("exclusive"). To indicate no limit, select **open**.

9.6 Enter the default value for the attribute in the Default value field. Check the **Auto assign value** checkbox to automatically assign the default value to the attribute when a new instance of its parent object is created.

9.7 Enter the units for the attribute value in the Units field.

9.8 Use the Symbol map table to assign symbols ("friendly" names) to specific values that the attribute can have.

- Symbol Map table—Lists the names and values of defined symbols.

- New Symbol field—Use to add a symbol to the Symbol Map table. Type the name in this field and click the **Add** button.

- **Add** button—Adds the new symbol named in the New Symbol field to the Symbol Map table.

- **Allow other values** checkbox—If checked, allows the attribute to have values other than those specified in the symbol map table. If unchecked, only the values in the symbol map table are allowed for the attribute.

- **Delete** button—Deletes the selected symbol from the Symbol Map table.

9.9 Describe the attribute to users of a model in the Comments field. Comments should include a definition of the attribute, how it is used by an object, and any other information helpful to the model user.

9.10 Click **OK** to accept the current property values and close the Properties dialog box, or **Cancel** to close the dialog box without making any changes to the property values.

10 Click **OK** to save the information, close the Extended Attributes dialog box, and return to the Attributes dialog box.

11 Click **OK** to accept the attributes values and close the Attributes dialog box, or **Cancel** to close the dialog box without making any changes to the attributes.

End of Procedure 3-1

Editing Attributes for Many Objects

You might need to set values for the same attributes on several objects in a network model. For a few attributes, you could make these changes individually using Procedure 2-1. However, this method can be tedious and error-prone for large numbers of objects. NETWARS provides several alternate methods for quickly changing attributes on many objects, one of which is using the **Edit Objects Using Template** command on the **Edit** menu.

NETWARS allows you to edit attributes for multiple objects in one window, using an attribute table similar to a User Defined Report. You can create your own *attribute templates* (as described in Procedure 2-3) to generate tables of specific attribute values for one or more objects in a network. An attribute template specifies the following information about an attribute table:

- Table category and name—Every attribute table belongs to a category, such as "IP" or "Application", and has a name that identifies it.

- Object types to include—A table can include all network objects, a specific type of network object, or objects selected by custom criteria that you specify.

- Attributes to include—Each row of an attribute template specifies the title and contents of a column that will appear in the attribute table.

Procedure 3-2   Edit Attributes For Many Objects Using a Template

1   Select **Edit > Edit Objects Using Template.** The Edit Objects Using Template dialog box displays.
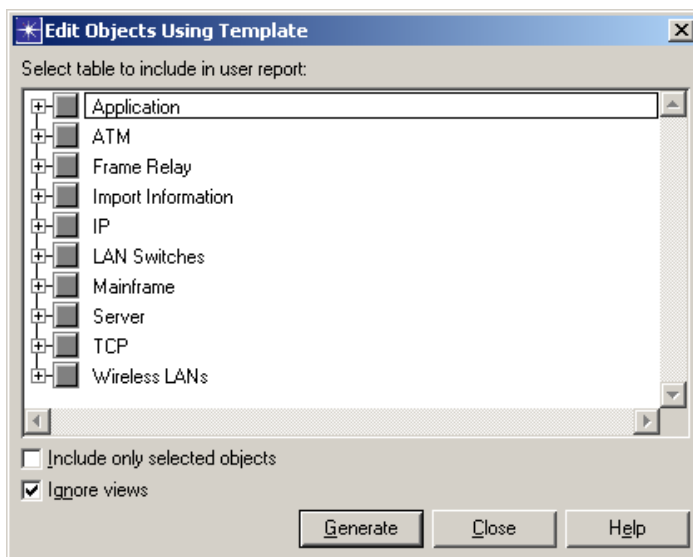
Figure 3-6   Edit Objects Using Template dialog box

2   Select an attribute template in the Select table to include in user report list box from which to generate an editable table of attribute values.

To select an entire category of tables, click on the top-level category entries. To select an individual table, expand the top-level category subtree and click on the table entry. Selected tables and categories show a green checkmark. Categories with some but not all of its tables selected will display a green dot instead.

Note—If you want to define a custom template, click **Close**, define a template as described in Procedure 2-3, then begin this procedure again.

3   Select the **Include only selected objects** checkbox if you want to collect table data only for objects that are currently selected in the network.

4   Select the **Ignore views** checkbox if you want to collect table data even for objects that are not in the current view or views of the network.

5   Select the **Send reports to the Report Server (*host*)** checkbox if you want to send reports to the Report Server. This option is active only when a Report Server license is available.

   5.1   Specify a name for the report in the Report Server text field.

6 Click **Generate** to generate the table data for all of the selected tables. The corresponding dialog box displays.

	Router Name	Router AS Number	Neighbor IP Address	Neighbor AS Number	Address Family	EBGP Multihop
1	cisco7000	Auto Assigned			IPv4	
2	NIPR router	Auto Assigned			IPv4	
3	SIPR router	Auto Assigned			IPv4	
4	cisco4500	Auto Assigned			IPv4	
5	IP_cloud	Auto Assigned			IPv4	
6	cisco4500	Auto Assigned			IPv4	
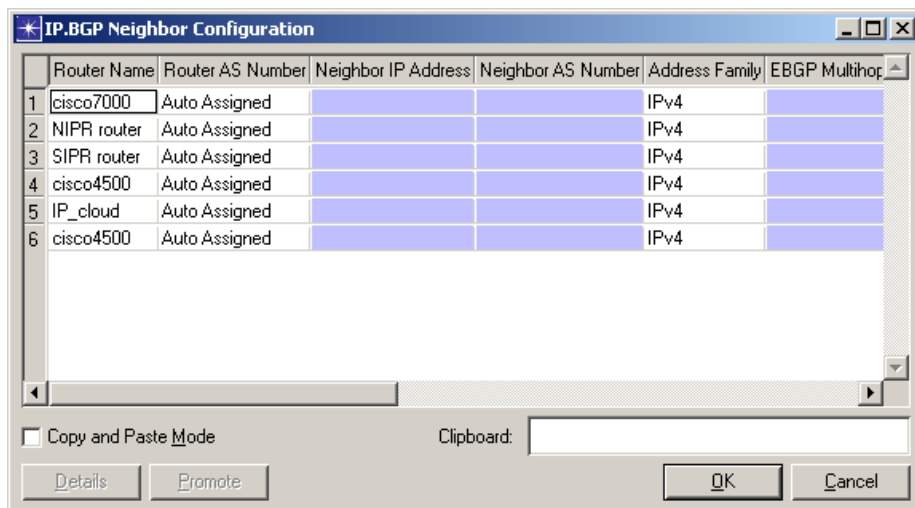
Figure 3-7   Table of Generated Attributes

7 Edit the attributes as needed. There are two editing modes:

7.1 Standard Mode—Click on any attribute value and edit it as you would in the Attributes dialog box.

7.2 Copy and Paste Mode—Copy an attribute value and paste it to one or more similar attributes. This mode disables the standard methods for changing attribute values (edit fields, pull-down lists, and so on).

- Select the **Copy and Paste Mode** checkbox.

- Select an attribute value and press **<Ctrl>**+**<C>**.

- Select the attribute value or a range of contiguous values and press **<Ctrl>**+**<V>**.

   You can paste only into the same attribute column from which you copied.

8 When you are finished editing attribute values, click **OK** to make the changes.

Procedure 3-3   Define or Edit an Attribute Template

1 Select **Edit > Edit Attribute Template.** The Edit Attribute Template dialog box displays.
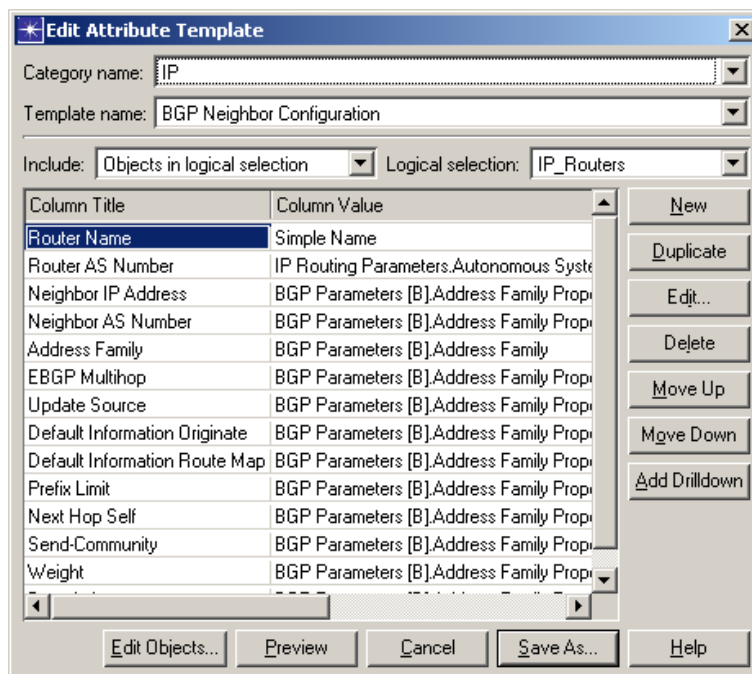
Figure 3-8   Edit Attribute Template dialog box

Each table row corresponds to an attribute column in tables generated from this template. At first, the only column defined is for an object name.

2 Choose whether to define a new template or edit an existing one:

2.1 To define a new template, leave the category and template names set to "unnamed". (You will specify these names when you save the template.)

2.2 To edit an existing template, select its category and name from the pull-down menus.

3 Choose what object types to include in the table:

3.1 Select an object type from the Include pull-down menu.

3.2 If you selected "Objects in logical selection", use the Logical selection pull-down menu to specify a selection criterion.

4 Choose what attributes to include in table columns:

4.1 To add a new undefined column, click **New**.

4.2 To add a new column based on an existing one, select the desired attribute row and click **Duplicate**.

4.3 To edit a column definition, select a row and click **Edit**. (Alternatively, you can double-click the Column Value field.) This opens a dialog box for specifying the contents and format of the table column.

4.4 To delete a column, select its row and click **Delete**.

4.5 To add a drilldown, select a row and click **Add Drilldown**. A drilldown is a child table that you open by following a link in the parent table. Drilldowns are useful when you want to show a set of related attributes in a separate window. For example, you might want to create a drilldown table for IP port attributes on a router.

This operation creates a drilldown table definition, which is initially equivalent to the current table definition. To edit the drilldown table, select the row and click **Edit**.

5 Move attributes around as needed to modify the appearance of tables generated from the template:

5.1 To change the sequence of attributes, select an attribute row and click **Move Up** or **Move Down** (thus moving the resulting table column left or right, respectively).

5.2 To see what the generated table will look like, click **Preview**. A dialog box displays a preview of the table to be generated using the current template.

Figure 3-9   Preview of Table to be Generated from Template

6 Click **Save As** in the Edit Attribute Template dialog box.

6.1 If you are defining a new template, enter names for the category and table. The category can be one of the existing categories or a new one.

6.2 If you are editing a template, you can either keep the original category and table names (to replace the original template) or change the table name (to create a new version of the original).

The template is saved with the extension `.urep.xml`. New templates are saved in your default model directory; edited templates are saved in the same directory as the original template.

7 Click **Cancel** to close the dialog box.

End of Procedure 3-3

Configuring Protocols

Each protocol supported by NETWARS and listed under the Protocols menu has commands associated with it that enable you to perform operations related to the configuration of that protocol.

The protocols are not documented in this manual; please refer to the OPNET documentation listed for each menu option listed below (consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**.)

Procedure 3-4   Access Options for Configuring Protocols

1 Select **Protocol > Applications > *option***. Select from the provided options to add ACE application models to the existing network, configure application demands, or clear various types of application traffic from the network. Refer to the *Applications Model User's Guide* for more information (available via **Protocol > Applications > Model User Guide**.)

2 Select **Protocol > IP > *option***. Select from the provided options to assign, configure, and clear IP addresses, configure routing protocols, enable reachability analysis to view routing data from Flow Analysis or DES, characterize traffic demands, display/hide routes, ping traffic, and configure QoS. Refer to the *IP Model User's Guide* for more information (available via **Protocol > IP > Routing > Model User Guide**.)

3 Select **Protocol > BGP > *option***. Select from the provided options to configure BGP start time, status, IBGP and EBGP peers, and route redistribution from other protocols into BGP. Refer to the *BGP Model User's Guide* for more information (available via **Protocol > BGP > Model User Guide**.)

4 Select **Protocol > EIGRP > *option***. Select from the provided options to configure EIGRP start time and route redistribution from other protocols into EIGRP. Refer to the *EIGRP Model User's Guide* for more information (available via **Protocol > EIGRP > Model User Guide**.)

5 Select **Protocol > IGRP > *option***. Select from the provided options to configure IGRP start time and route redistribution from other protocols into IGRP. Refer to the *IGRP Model User's Guide* for more information (available via **Protocol > IGRP > Model User Guide**.)

6 Select **Protocol > IS-IS > *option***. Select from the provided options to configure interface metrics, interface circuit types, system types, metric styles, SPF calculation parameters, and route redistribution. Refer to the *IS-IS Model User's Guide* for more information (available via **Protocol > IS-IS > Model User Guide**.)

7 Select **Protocol > OSPF > *option***. Select from the provided options to configure interface cost and timers, OSPF start time, areas, and route redistribution. Refer to the *OSPF Model User's Guide* for more information (available via **Protocol > OSPF > Model User Guide**.)

8   Select **Protocol > RIP > *option***. Select from the provided options to configure RIP start time, and route redistribution from other protocols into RIP. Refer to the *RIP Model User's Guide* for more information (available via **Protocol > RIP > Model User Guide**.)

9   Select **Protocol > RSVP > *option***. Select from the provided options to configure interface status. Refer to the *RSVP Model User's Guide* for more information (available via **Protocol > RSVP > Model User Guide**.)

10   Select **Protocol > ATM > *option***. Select from the provided options to configure over-subscription on a per-service class basis for all or selected nodes in the network, display/hide ATM VC routes, and create IMA groups. Refer to the *ATM Model User's Guide* for more information (available via **Protocol > ATM > Model User Guide**.)

11   Select **Protocol > Frame Relay > *option***. Select the provided option to configure PVCs between nodes in the network. Refer to the *Frame Relay Model User's Guide* for more information (available via **Protocol > Frame Relay > Model User Guide**.)

12   Select **Protocol > VLAN > *option***. Select from the provided options to configure VLANs for selected nodes or links, configure selected links as trunk links, and enable/disable VLANs for selected switches. Refer to the *VLAN Model User's Guide* for more information (available via **Protocol > VLAN > Model User Guide**.)

13   Select **Protocol > STP > *option***. Select from the provided options to apply or clear Spanning Tree visualization.

End of Procedure 3-4

Using Configuration Utilities

NETWARS provides a Configuration Wizard to help you configure Promina and Multiplexer utility nodes. Utility nodes contain information/attributes pertaining to multiple nodes in the scenario. They provide a convenient way of configuring information that is conceptually shared.

A sample Configuration Wizard workflow for configuring Prominas is provided below. Workflows are similar for other utility nodes; however, they do vary depending on the utility being configured.

Procedure 3-5   Configure Promina Devices

1 Select **Topology > Configuration Utilities > Promina** to start the wizard. The scenario must contain at least two Promina devices that have available LAN port to support the configuration; otherwise, an error will be prompted and the wizard is terminated.

2 Select the source Promina device and its associate LAN port. To help identify the LAN port, the subscriber side device is also shown in the drop-down menu.
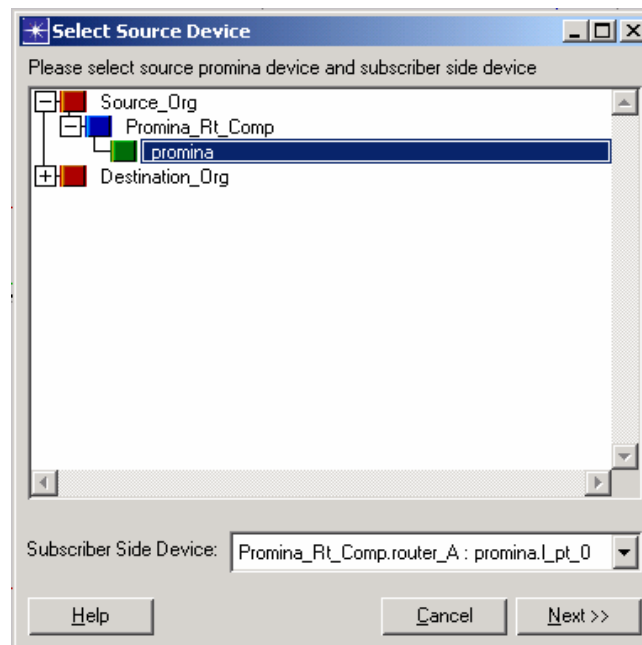

Figure 3-10   Selecting a Source Promina Device

The treeview contains all Promina devices in the scenario that have available LAN port to support the configuration. Expand the treeview and select the desired source device by left-clicking on the Promina device.

3 Once the source device is selected, the Subscriber Side Device drop-down list becomes enabled. This drop down list provides the available LAN ports on the selected promina device. It consists of the <subscriber side device>: <LAN port that's connected to the subscriber side device>. Select the desire LAN port from the list, and click the **Next>>** button to advance to the next step.

3.1 If you want to cancel out from the wizard, click **Cancel**.

3.2 If you want to return to the previous step, click **<<Back**.

4 After the source Promina device has been selected, select the destination Promina device and its LAN port. This step is the same as selecting the source Promina device described above. Only those Prominas that have LAN side devices connected to them are considered to be potential source/destination Promina devices.

5 After the source and destination Promina devices and ports have been selected, configure circuit attribute values including the CCSD codes for the Promina circuit name as shown in the figure below. All fields have default values.

Figure 3-11   Promina Circuit Attributes dialog box

The following attributes can be set for the Promina circuit:

- **Direction –** The direction of the circuit; some ports can only receive and some only send. **Dual** represents a full duplex circuit.

- **Max Port Speed –** Ports on the same card can have different maximum speed values; 64Kbps is the only value allowed for PRI-type cards. The **Auto-sense** option will take the link data rate as the circuit speed.

- **Call Type –** A **permanent** call will be set up at the beginning of simulation and never torn down; a **demand** call will be set up whenever demanded by the connected end devices and torn down if inactive.

- **Inactivity Timer –** If there is no packet exchange for the specified amount of time, the demand call will be torn down.

- **Start/Stop Time –** The start/stop time for the Promina circuit.

- **Number of Circuits –** Specify one or more circuits over the same path.

- **Define Circuit Path –** If no other Promina devices exist in the scenario, other than the selected source and destination Promina devices, the **Define Circuit Path** checkbox will be unchecked and disabled; you can click the **Complete** button to complete the Promina circuit configuration. If another Promina does exists, this field will be checked by default; you can click the **Next>>** button to add more Prominas to the circuit sequence.

  Click the **Help** button for information on the current dialog box, **<<Back** button to return to the previous dialog box, and **Cancel** button at anytime to terminate the wizard.

6 If the Define Circuit Path checkbox is unchecked, click the **Complete** button to complete the Promina circuit configuration and circuit information is automatically applied to the utility OPFAC that contains the pro_portmap_utility node. If the Define Circuit Path checkbox is checked, click the **Next>>** button to specify path for the circuit as shown below.
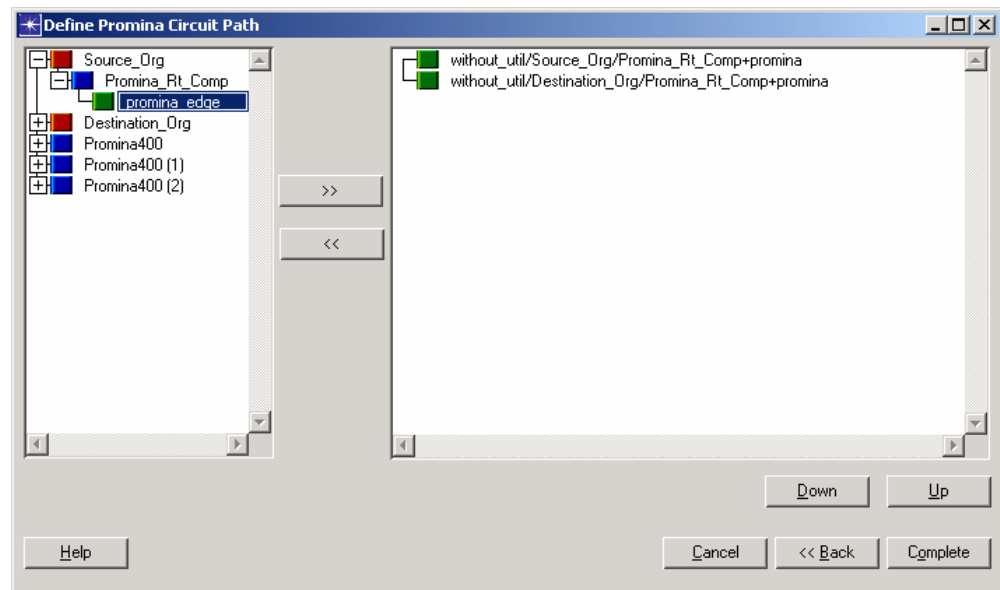

Figure 3-12   Define Promina Circuit Path dialog box

The left-side treeview of the Define Promina Circuit Path dialog box contains all Promina devices in the scenario with the exception of the source and destination Promina devices. The right-side treeview indicates the path of the circuit. Initially, it lists only the source and destination Promina devices.

7 Expand the left-side treeview and select an intermediate Promina device and click the **>>** button to add it as an intermediate node to the right-side treeview. Intermediate devices are positioned before the destination device so devices are added in sequence. Select an intermediate node from the right-side and click the **<<** button to remove it from the path. The removed device will be added back to the left-side treeview and becomes available for later selection.
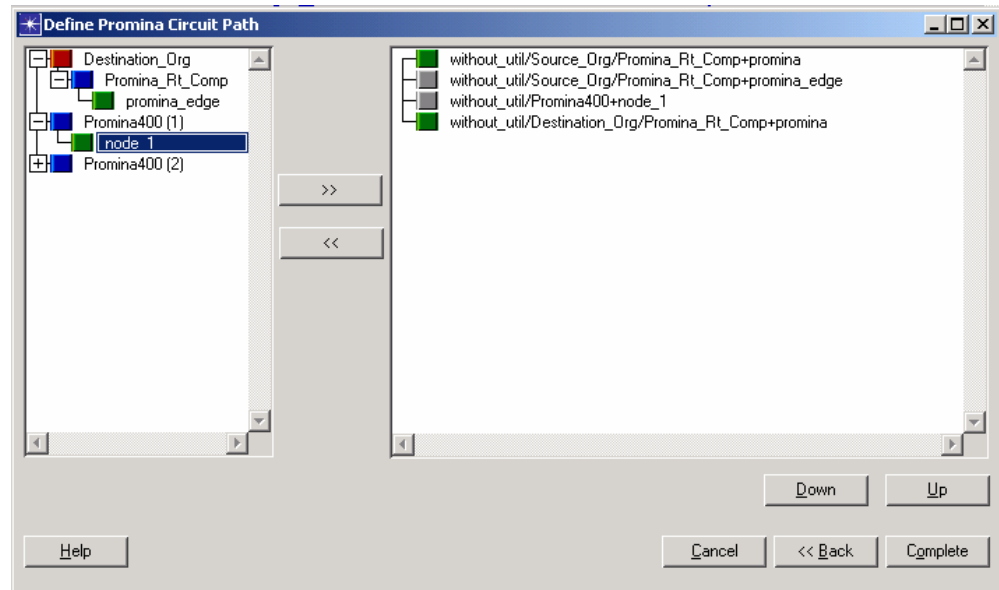
Figure 3-13   Defining Intermediate Nodes

Reposition intermediate nodes if needed by first selecting the desired device and clicking on the **Down** or **Up** buttons to move it down or up, respectively, from the path.

8 Click the **Complete** button to finish the configuration and apply the information to the utility OPFAC that contains the Promina_Utility node.

The Promina circuit definition is maintained on the Promina_Utility node, which resides in the Multiplexer Utility OPFAC. You can view the attributes of the circuit by accessing the **Attributes** of the **Promina_Utility node** and clicking on the **Value** cell of the **Portmap Configuration** attribute.

9 Once the configuration is completed, users have the opportunity to configure another Promina circuit without exiting the wizard through a pop-up dialog box that displays.

10 Click the **Help** button for information on the current dialog box, **<<Back** button to return to the previous dialog box, and **Cancel** button at anytime to terminate the configuration wizard.

End of Procedure 3-5

Using the Model Assistant

The Model Assistant allows you to automate frequently applied scenario changes by editing and saving data in model assistant files and then applying the data to your network. A model assistant file is an ASCII text file that specifies a set of changes to a scenario. You can specify multiple changes in one file, and then apply the file to make multiple changes in one operation. For example, you can create a model assistant file to:

- Create/remove objects (subnetworks, nodes, links, and demands),
- Modify object attributes, and
- Specify object locations.

Procedure 3-6   Edit a Model Assistant File

1   Select **Topology > Model Assistant > Edit File**. The Edit Model Assistant File dialog box displays.
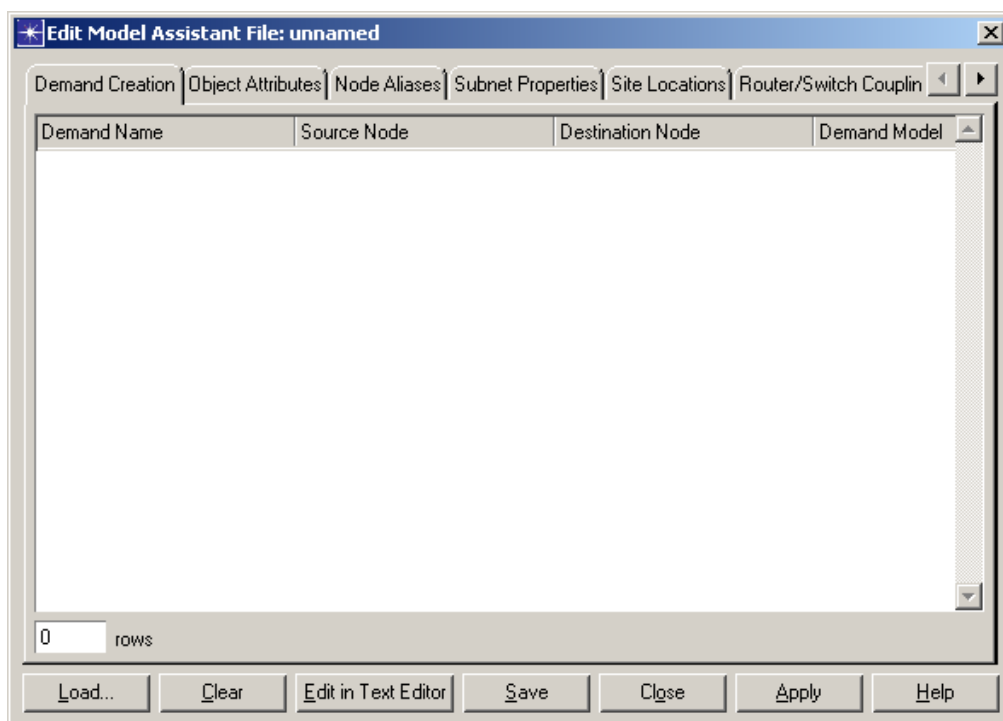
Figure 3-14   Edit Model Assistant File dialog box

This dialog box contains multiple tabbed pages; each page has fields for entering the data you want to apply.

2   Specify the information you want to apply on each page (click the tabs to view specific pages).

2.1   If you do not want to apply a certain type of data, leave the page empty.

2.2   If you want to edit an existing file, click **Load**, choose the file to load, and click **OK**. The dialog box updates each tabbed page to reflect the data in the loaded file.

2.3 If you want to reset all settings on all pages and reenter your data, click **Clear**.

2.4 If you want to open the model assistant file you are editing in the text editor specified in the editor_prog preference, click **Edit in Text Editor**. This gives you some additional flexibility when editing your model assistant file. When the text editor is closed, the changes you made are immediately reflected in the GUI.

3 Click the **Demand Creation** tab if you want to specify demands to create in the network. Each row in the table represents a demand to be created.

3.1 Type the name of the demand object in the Demand Name field.

3.2 Enter the source node of the new demand in the Source Node field and the destination node of the new demand in the Destination Node field. You do not need to specify the full hierarchical names of these nodes. If there is only one node with the specified name in the network, the model assistant will find it. You can select a node model from the pull-down menu (single click) or type the model name (double click). The pull-down menu lists all nodes in the network and in the Node Creation page.

3.3 Enter the model name of the demand object in the Demand Model field. You can select a demand model from the pull-down menu (single click) or type the model name (double click).

4 Click the **Object Attributes** tab if you want to specify attributes for the various objects in your network. Each row in the table represents an object attribute.

4.1 Enter the name of the object whose attribute you want to set in the Object Name field. You do not need to specify the full hierarchical name of this object. If there is only one object with the specified name in the network, the model assistant will find it. You can select a network object from the pull-down menu (single click) or type the object name (double click). The pull-down menu lists all objects in the network.

4.2 Enter the name of the attribute to set in the Attribute Name field. This can be a simple or a compound attribute. Note that simple attributes include simple (non-compound) attributes of compound attributes.

4.3 Enter a new value for the specified attribute in the Attribute Value field. If you specified a compound attribute for the name, you can enter only symbolic values.

Specifying Attributes in the Model Assistant:

If you want to set an attribute that is part of a compound attribute, you must enter the full hierarchical name of the attribute. The hierarchical name includes each parent attribute and the row number. The best way to verify the correct format for an attribute name is to do the following:

• Right-click on a network object and choose **Edit Attributes**.

• If the attribute is part of a compound attribute, expand the attribute tree until the attribute you want to set is visible in the "Name" column.

• Set the "Value" field to the value you want to specify in the Model Assistant.

• Rest the cursor over the Value field. The tooltip shows the full hierarchical name and current value setting for the attribute.

5 Click the **Node Aliases** tab if you want to specify aliases on the nodes in your network. Each row in the table represents an alias.

    5.1 Enter the name of the node whose aliases you want to set in the Node Name field. You do not need to specify the full hierarchical name of this node. If there is only one node with the specified name in the network, the model assistant will find it. You can select a node from the pull-down menu (single click) or type the node name (double click). The pull-down menu lists all nodes in the network.

    5.2 Enter an alias to set on the specified node in the Alias field.

6 Click the **Subnet Properties** tab if you want to set certain display properties on subnets (OPFACs/Organizations) in your network. Each row in the table represents a property.

    6.1 Enter the name of the OPFAC or Organization whose properties you want to set in the Subnet Name field. You do not need to specify the full hierarchical name of the OPFAC or Organization. If there is only one OPFAC or Organization with the specified name in the network, the model assistant will find it. You can select an OPFAC or Organization from the pull-down menu (single click) or type the OPFAC or Organization name (double click). The pull-down menu lists all OPFACs/Organizations in the network.

    6.2 Select the OPFAC or Organization property to set in the Subnet Property field. You have five choices: Display Resolution, Display Units, Grid Draw Style, Grid Division, and Map. Note that none of the "properties" that appear when you right-click on the OPFAC or Organization and select **Edit Attributes** are on this list. This is because those are the OPFAC or Organization's attributes, and can be set using the **Object Attributes** tab.

    6.3 Enter the value of the OPFAC or Organization property you want to set in the Property Value field. The input to this column depends on your choice of property in the previous column as follows:

        • Display Resolution — Display resolution of your OPFAC or Organization in pixels per whatever your OPFAC or Organization units are.

        • Display Units — Units in which your OPFAC or Organization will be displayed. Your choices are miles, feet, kilometers, meters, and degrees.

        • Grid Draw Style — Grid draw style in the OPFAC or Organization. Your choices are solid, dashed, or disabled.

        • Grid Division — Grid division in the OPFAC or Organization, in whatever your display units are.

        • Map — Background image for the OPFAC or Organization. The value specified must be a valid image file.

7 Click the **Site Locations** tab if you want to set the geographical locations of sites in your network. Each row in the table represents a site location.

    7.1 Enter the name of the site whose location you want to set in the Site Name field. You do not need to specify the full hierarchical name of this site. If there is only one site with the specified name in the network, the model assistant will find it. You can select a site from the pull-down menu (single click) or type the site name (double click). The pull-down menu lists all sites in the network.

7.2 Select the format in which your site's location is to be specified in the Location Type field. You have four choices: [NPA, NXX], [City, Country], [City, Province (Canada)], [City, State (USA)], [Lat, Long] and [X, Y (in parent's units)].

7.3 Enter the location of the site in the Location Data field. The input to this column depends on your choice of format in the previous column as follows:

- NPA, NXX — Area code (NPA) and first three digits of the phone number (NXX) of the area where your node is to be placed (e.g. "240, 497"). This format is only valid for locations within the US and Canada.

- City, Country — City and country where your node is to be placed (e.g. "Paris, FR"). You can specify the country using the full country name or a two letter country code from the ISO-3166 standard.

- City, Province (Canada) — City and province where your node is to be placed (e.g. "Toronto, ON"). You can specify the province using a two-letter code, or the full province name. This format is only valid for locations within Canada.

- City, State (USA) — City and state where your node is to be placed (e.g. "Bethesda, MD"). You can specify the state using a two letter code, or the full state name. This format is only valid for locations within the US.

- Lat, Long — Latitude and longitude where your node is to be placed (e.g. "38.4, -75.2").

- X, Y (in parent's units) — X and Y coordinates at which your node is to be placed (e.g. "1234, 234.5"). The values of the coordinates you specify will be interpreted in the units of the site's parent OPFAC or Organization.

8 Click the **Router/Switch Coupling** tab if you want to couple routers with switches in your network. Each row in the table represents a coupling.

8.1 Enter the name of the router to be coupled with a switch in the Router Name field.

8.2 Enter the name of the switch to be coupled with the chosen router in the Switch Name field.

9 If you want to save your data to a model assistant file so you can reuse it later, click **Save**. If an existing file is selected, it will be overwritten.

10 Click **Apply** to apply the data in the various tabs of the dialog box to the current network, or **Close** to close the dialog box without applying your data or saving any changes.

End of Procedure 3-6

Procedure 3-7   Apply a Model Assistant File to the Network

1   Select **Topology > Model Assistant > Apply File**. The Select a Model Assistant File to Apply dialog box displays.
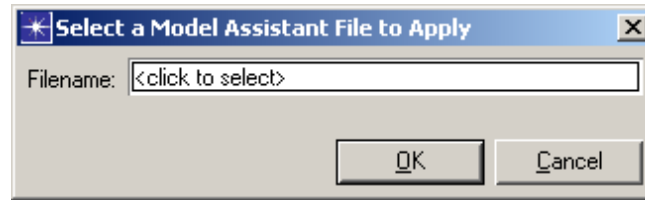

Figure 3-15   Select a Model Assistant File to Apply dialog box

2   Enter the name of the Model Assistant file that you want to apply to the current network in the Filename field, and then click **OK**.

End of Procedure 3-7

4   Visualization and Navigation

NETWARS provides several features that make it easier and faster for you to visualize and navigate your network model. Features discussed in this chapter can help you to find and edit objects, view and edit demands, visualize routing protocols and link loads, size and scale icons, set display preferences, and visualize the effects of changing the network time.

Finding Objects by Attribute

NETWARS provides a Selection Filter feature that enables you to find and select objects quickly and easily based on their attributes.

Procedure 4-1   Find Objects by Attribute

1   Select **Edit > Select**. The Selection Filter dialog box displays.

2   At the top of the Selection Filter dialog box, select the type of object that you want to find by clicking the appropriate **Object Type** check box.

3   In the table in the middle of the Selection Filter dialog box, click in the top empty field of each column to display a drop-down menu to use for selecting objects. Menu options change dynamically with the type of object chosen for selection.

    From these menus, select one or more attributes and their matching criteria to be used for selecting objects:

    3.1   In the **PROP** column, select **Require** if you want only to find objects that satisfy *all* attribute criteria for selection, or select **Consider** if you want to find any objects that satisfy at least *one* of the attribute criteria for selection.

    3.2   In the **Attr Name** column, select the name of the attribute to be used as the selection criteria.

    3.3   In the **?=** column, select the type of matching to be used.

       • For a string, the only option is a complete match or **equal to**.

       • For numeric values, the satisfying condition could be **less than**, **greater than**, or **equal to** the specified value.

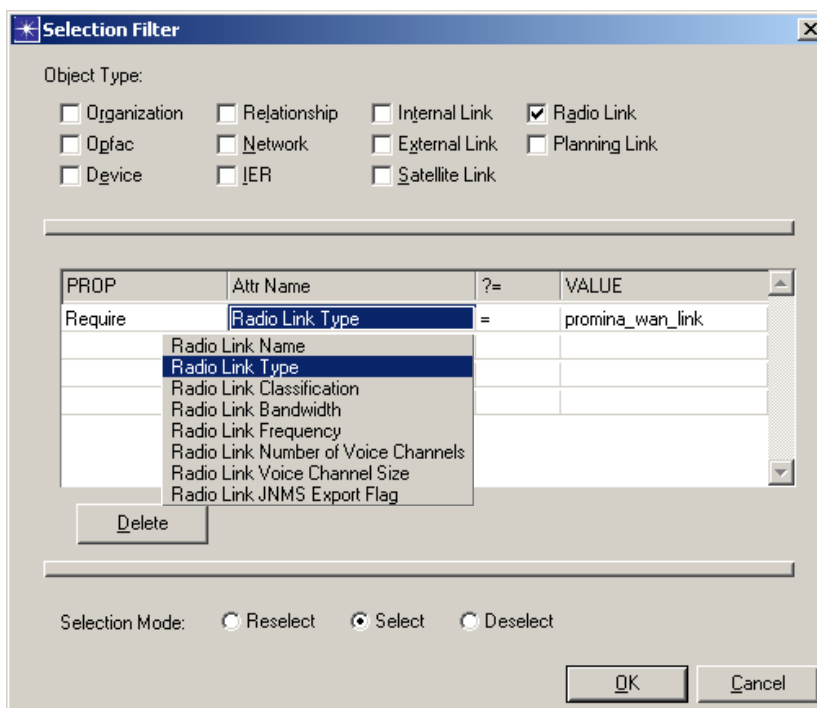**3.4** In the **VALUE** column, select the value that the attribute must match for selection.

Figure 4-1   Searching for Objects by Attribute

4 Click **OK**. NETWARS searches through all the objects in the scenario looking for those that match the specified criteria. When it finds objects that match the given criteria, it marks those objects as selected.

End of Procedure 4-1

Finding Objects by Name

NETWARS provides a Find Node/Link feature that enables you to find and select objects quickly and easily based on their names.

Procedure 4-2   Find Objects by Name

1 Select **Edit > Find Node/Link** (or press **<Ctrl>**+**F**). The Find Node/Link dialog box displays.

2 In the **Name Contains** field, type a text string to search for specific names. You can specify part or all of the name. Leave the field blank to find all objects in the network.

3 Click **Find**. NETWARS searches through all the objects in the scenario looking for those that match the specified name. When it finds objects that match the given name, it displays those objects in the Results table by object name, name of the subnet that contains the object, and type of object.
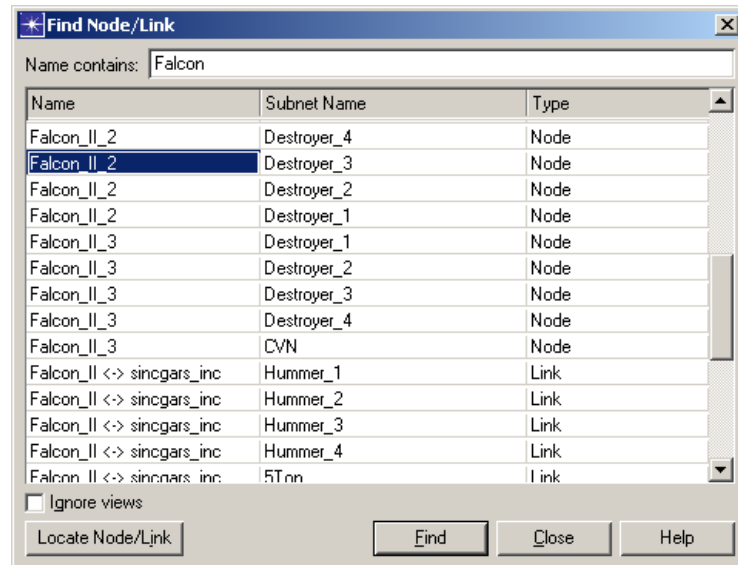

Figure 4-2   Searching for Objects by Name

4 To view a single found object in the workspace, select the object in the Results list and then click **Locate Node/Link**.

End of Procedure 4-2

Using the Network Browser

Large, complex network models can be difficult to navigate. The NETWARS Network Browser provides a treeview that you can use to organize and display objects (organizations, OPFACs, devices, links, etc.) in your network. This browser makes it much easier to find objects and edit their attributes, especially in large networks with multiple layers of subnets.

Procedure 4-3   Use the Network Browser

1 In Scenario Builder, select **File > Open Project** and then select an existing project to display.

2   Select **View > Show Network Browser** (or press **<Ctrl>**+**B**) to toggle open the Network Browser.
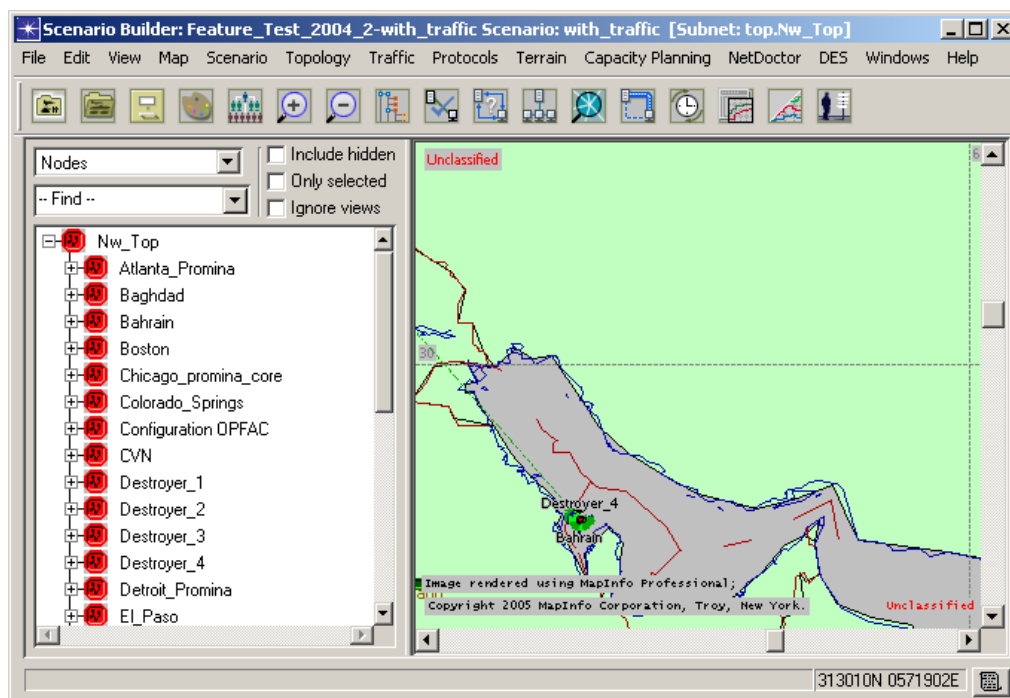

Figure 4-3   Open the Network Browser

The Network Browser pane displays on the left side of the workspace. The pane can be resized horizontally if desired.

3   Filter the object list as needed by using the two drop-down filters and/or three checkbox filters displayed above the object list:

3.1   The top drop-down list controls the type of objects that display in the object list. For example, the filter can be set to display nodes, links, IP devices, workstations, etc. The filter also organizes the list by the selected object. These objects appear as top-level children within each subnet; any connected objects appear as children of an object. For example, if the view is set to **Nodes**; the treeview then displays all nodes as top-level children within each subnet. Any objects connected to a node (such as links, demands or paths) appear as children of the node. If you select **All** from the drop-down menu, then all of the objects in the network display in the tree.

3.2   The bottom drop-down list is a search field that allows you to search for objects by name. To search for an object by name, type a text string in the **Find** field and then press **<Enter>**, or select a previously entered search term from the drop-down list. If corresponding objects are found, they are highlighted in the object list.

3.3   Check (or uncheck) the three checkbox filters to show (or hide) objects in the Network Browser:

- **Include hidden –** Check this box to show paths, demands and cell systems that are hidden as a result of settings in the View menu.

- **Only selected –** Check this box to show only those objects that are currently selected.

- **Ignore views –** Check this box to show objects that are excluded from the current network view.

4 Double-click an object in the list to display it in the workspace.
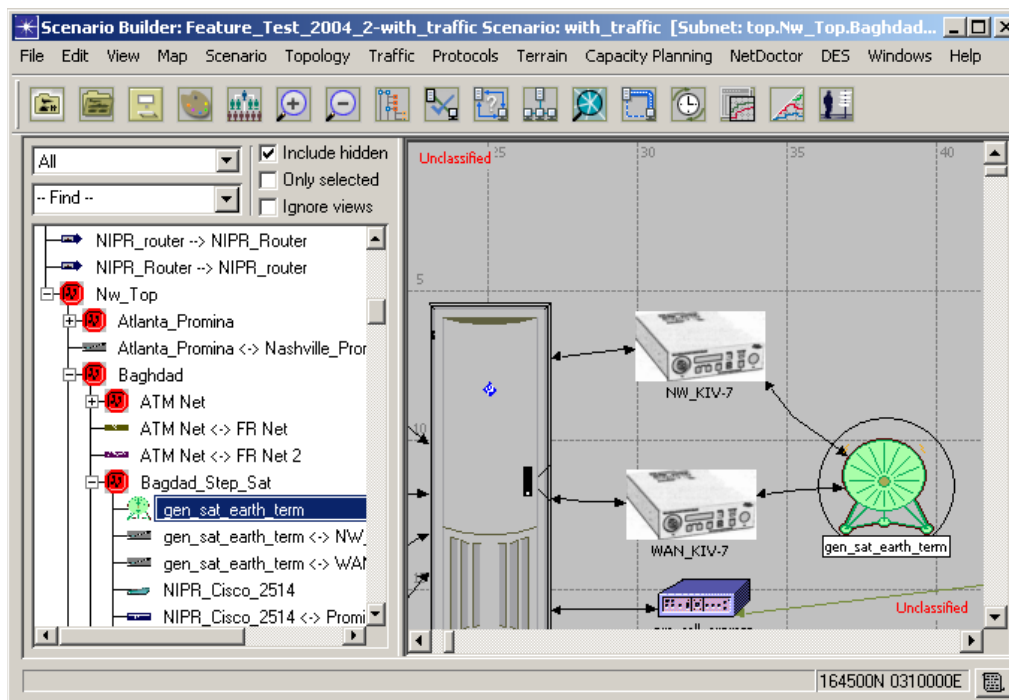

Figure 4-4   Display Object in the Workspace

5 Right-click an object in the workspace to display the standard shortcut menu for the object. Use the shortcut menu to edit object attributes, view similar nodes, choose simulation statistics, etc., quickly from the Network Browser.
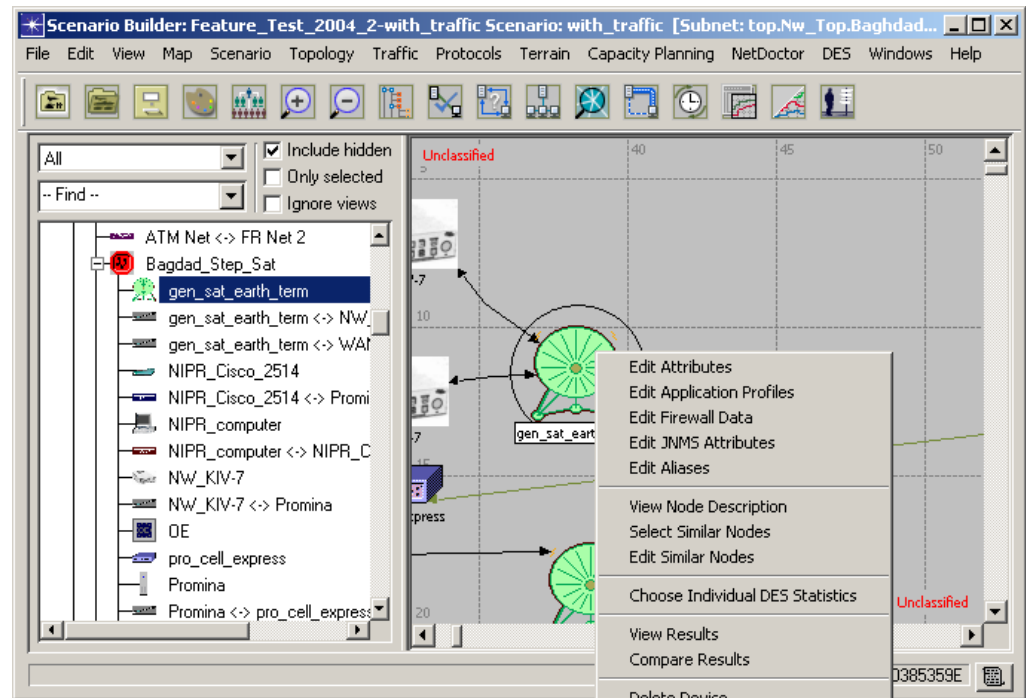
Figure 4-5   Access the Standard Shortcut Menu for an Object

6 Select **View > Show Network Browser** (or press **<Ctrl>**+**B**) again to toggle closed the Network Browser.

End of Procedure 4-3

Using the Flows Browser

The Flows Browser enables you to view and edit demands. Demands (as well as IERs) represent traffic in NETWARS. Demands shown in the Flows Browser are filtered by model and/or technology. Demands selectively appear when they or their attached nodes are selected in the treeview.

Procedure 4-4   Use the Flows Browser

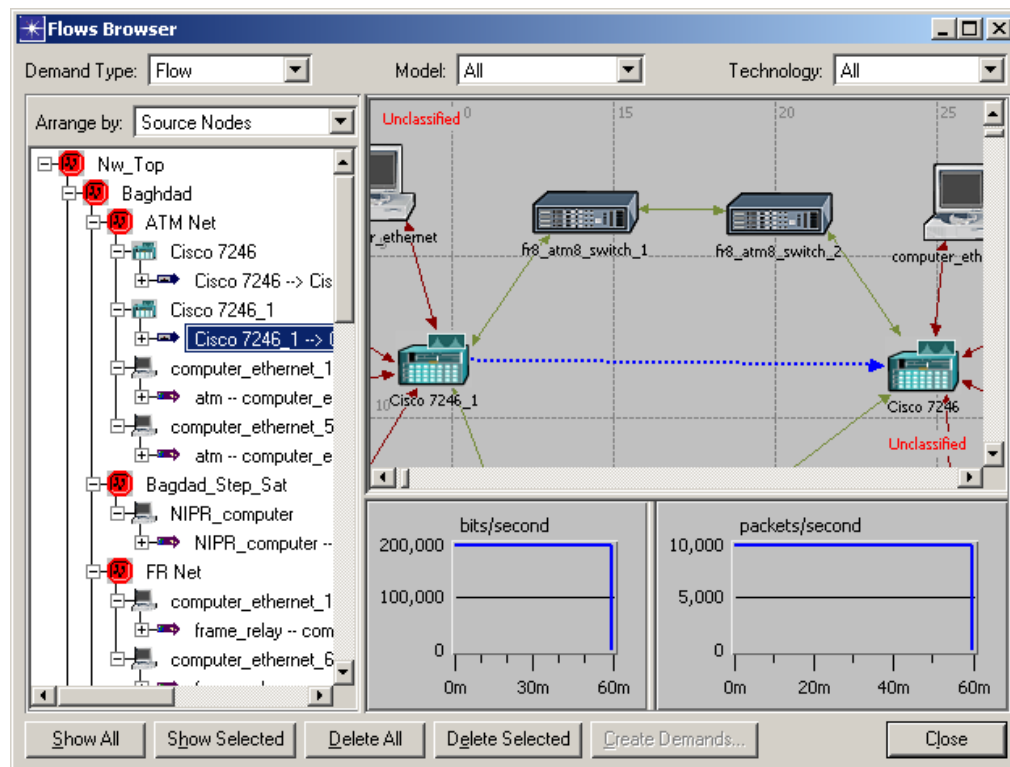1   Select **Traffic > Visualize > Open Flows Browser**. The Flows Browser dialog box displays.

Figure 4-6   Open the Flows Browser

Note—Right-clicking a demand in the treeview gives you access to the **Show/Hide Attached Demands** menu item.

2   Select a single demand in the treeview. Corresponding traffic graphs display to the right of the treeview below the workspace.

3 Right-click on a traffic graph and choose **Select Profile** to select a profile from the profile library to preview.
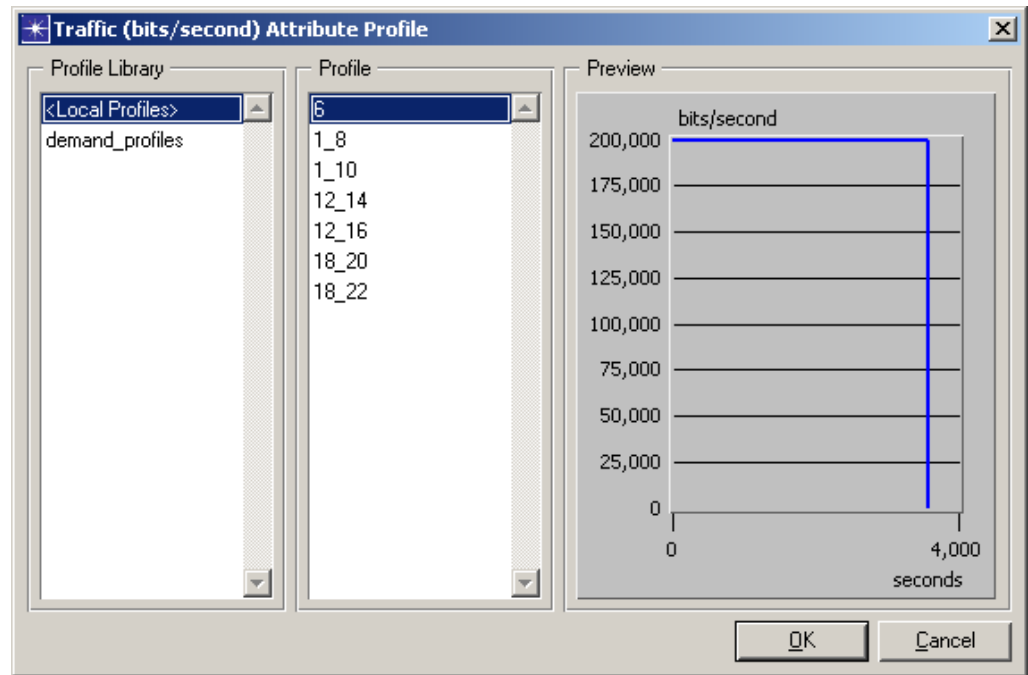

Figure 4-7   Traffic Attribute Profile dialog box

4 Right-click on a traffic graph and choose **Edit Profile** to edit the current profile's name, intervals, or start time (and show start time on the graph), or to import or export the profile from/to a text file.
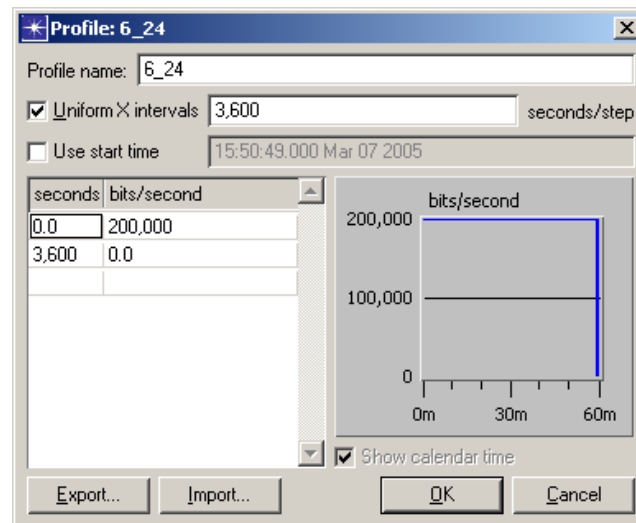

Figure 4-8   Edit Traffic Attribute Profile dialog box

End of Procedure 4-4

Visualizing Protocol Configurations

You can quickly determine which routing protocols are running on your network by using NETWARS' routing protocol visualization feature.

Procedure 4-5   Visualize Protocol Configurations

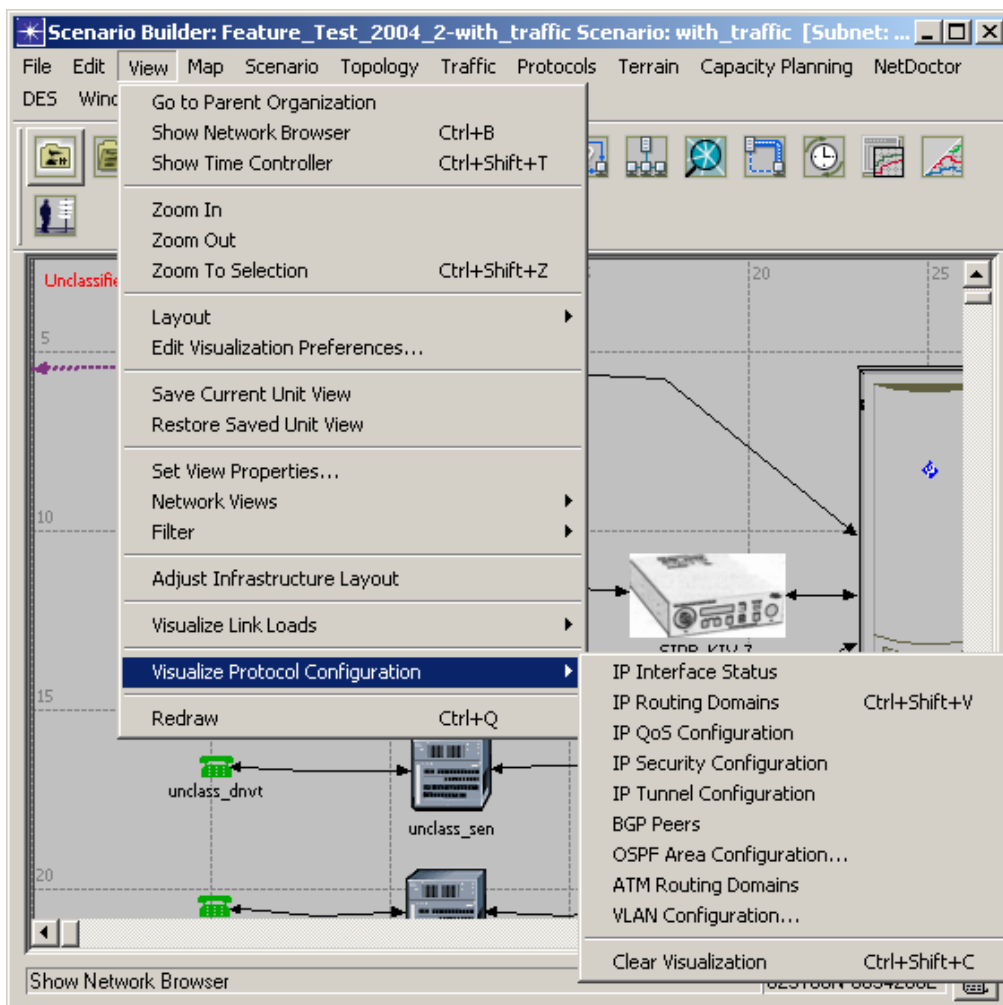**1** Select **View > Visualize Protocol Configuration > (*option*)**.

Figure 4-9   Protocol Visualization Menu Options

1.1 Select **View > Visualize Protocol Configuration > IP Interface Status** to show whether the connected interface is active or shutdown. A green up arrow indicates that the interface is active, whereas a red down arrow indicates that the interface is shutdown.
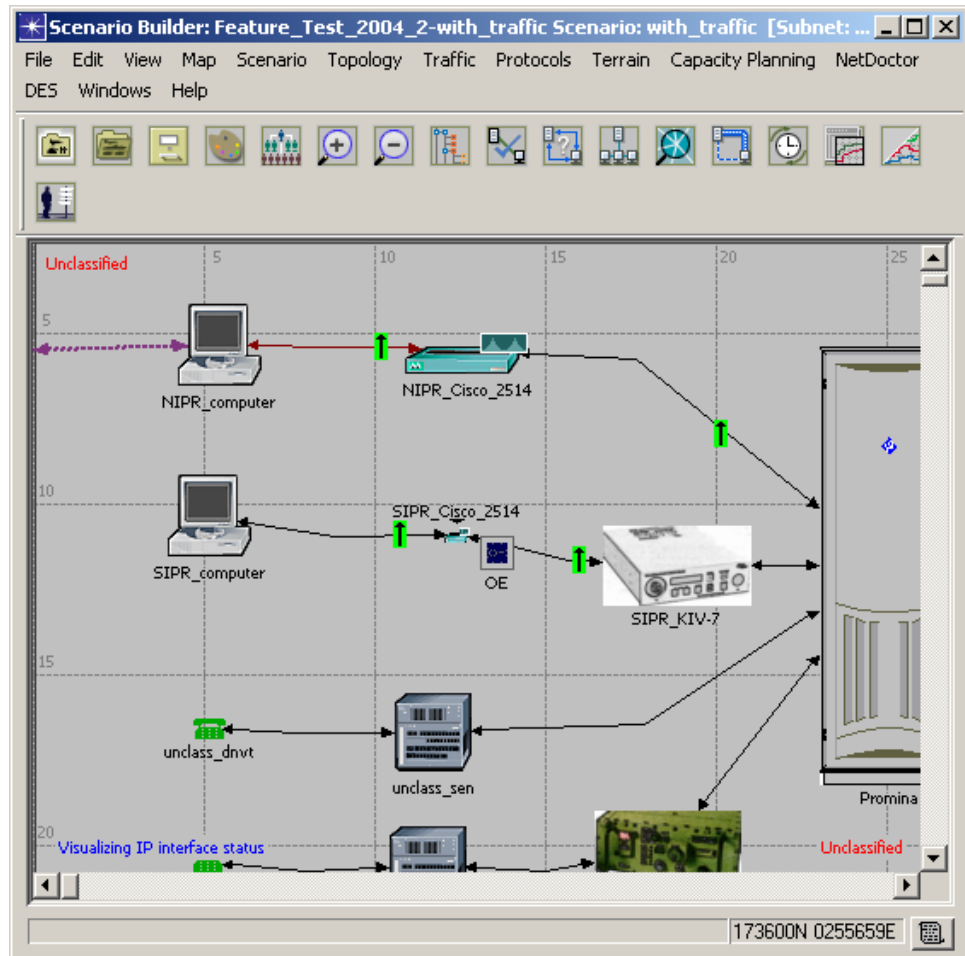

Figure 4-10   Visualizing IP Interface Status

Note—Use **<PageUp>** and **<PageDown>** keys to zoom out and zoom in.

1.2 Select **View > Visualize Protocol Configuration > IP Routing Domains** to show the routing protocols that are configured on the router interfaces.
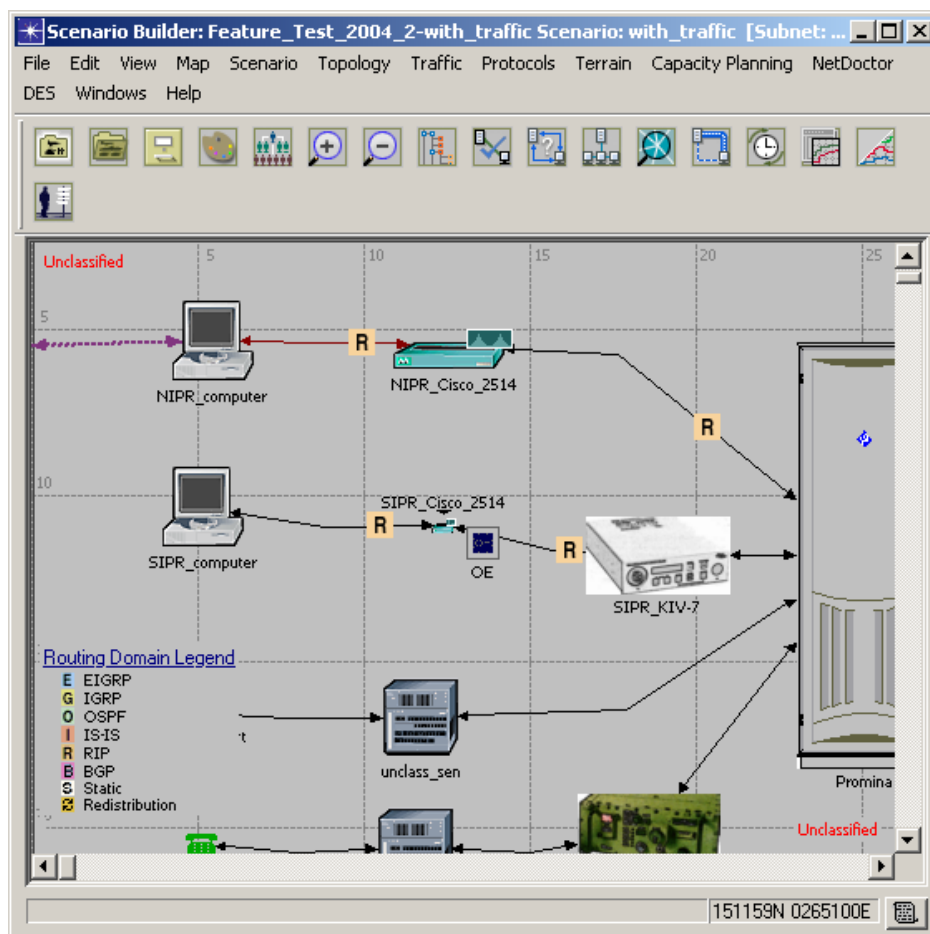

Figure 4-11   Visualizing IP Routing Domains

1.3 Select **View > Visualize Protocol Configuration > IP QoS Configuration** to show the IP QoS configuration on the links.

1.4 Select **View > Visualize Protocol Configuration > IP Security Configuration** to show the packet filtering security configured on the links.

1.5 Select **View > Visualize Protocol Configuration > IP Tunnel Configuration** to show the tunnels configured on the links.

1.6 Select **View > Visualize Protocol Configuration > BGP Peers** to show the BGP peering (neighbor) relationships in the network. EBGP confederation peers are shown in dark green, EBGP peers are blue, and all other colors indicate IBGP peers. You can view the tool tip of each peering visualization for more information about the peers. For networks using route reflection, the visualization also indicates if a router is a reflector or a client.

1.7 Select **View > Visualize Protocol Configuration > OSPF Area Configuration** to show the OSPF areas configured on the router interfaces.

1.8 Select **View > Visualize Protocol Configuration > ATM Routing Domains** to show which ATM routing protocols are configured on the switch ports.

1.9 Select **View > Visualize Protocol Configuration > VLAN Configuration** to show the VLANs configured in the network.

2 Select **View > Visualize Protocol Configuration > Clear Visualization** (or press **<Ctrl>**+**<Shift>**+**C**) to remove all protocol configuration visualizations from the workspace.

End of Procedure 4-5

Visualizing Link Loads

You can visualize the traffic loads on links in your network based on data collected from a Capacity Planning Evaluation. Link loads are easily identified in the workspace by link colors and line widths that you designate using the Link Load Visualization Settings dialog box. Link statistics are computed taking into consideration the entire duration of the Capacity Planning Evaluation, and values are reported in the workspace when you move your mouse over any link.

Note—You must first run a Capacity Planning Evaluation before you can visualize utilization. To run a Capacity Planning Evaluation, select **Capacity Planning> Evaluate**.

Procedure 4-6   Designate Link Load Visualization Settings

1 Select **View > Visualize Link Loads > Settings**. The Link Load Visualization Settings dialog box displays.
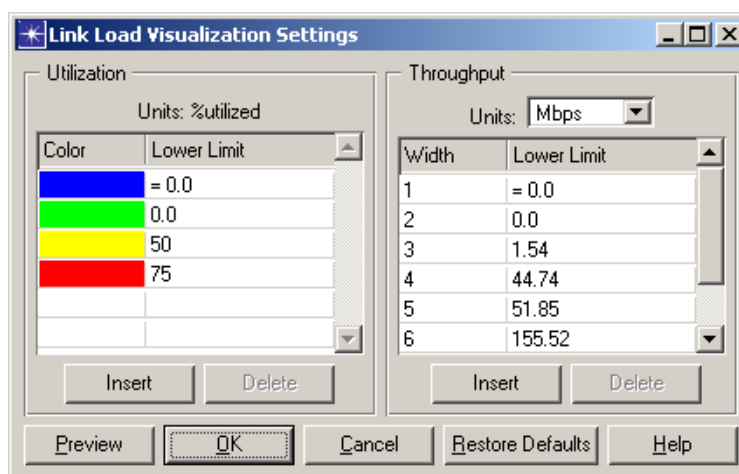

Figure 4-12   Link Load Visualization Settings dialog box

2 Control the display of link load link colors using the following options:

2.1 **Color** column - Lists the colors to use when visualizing link loads.

2.2 **Color Lower Limit** column - Shows the utilization value that corresponds to each color. The number represents the percentage of the link that is utilized.

2.3 Click **Insert** to add a new row to the color table.

2.4 Click **Delete** to delete the selected row from the color table.

3 Control the display of link load widths using the following options:

3.1 **Units** pulldown - Specifies the throughput units to use when interpreting the values in the width table.

3.2 **Width** column - Lists the link thickness to use when visualizing link loads.

3.3 **Width Lower Limit** column - Shows the throughput value that corresponds to each link thickness.

3.4 Click **Insert** to add a new row to the width table.

3.5 Click **Delete** to delete the selected row from the width table.

4 Click **Restore Defaults** at any time to reset the dialog box to the default link visualization settings.

5 Click **Preview** to apply the current settings to the network. This button only appears if link load visualization is enabled.

6 Click **OK** to save the specified settings and close the dialog box.

End of Procedure 4-6

Procedure 4-7   Visualize Link Loads

1 Select **View > Visualize Link Loads > Color by Link Load**. The Color Links by Load dialog box displays.
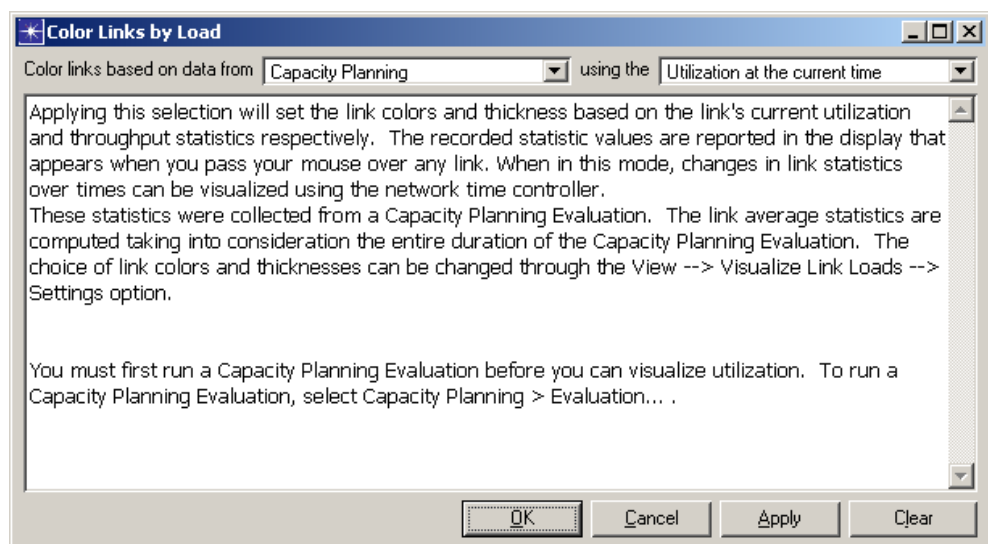

Figure 4-13   Color Links by Load dialog box

2 Set the link colors and line widths based on the link's peak, average, or current utilization (depending upon which one you choose in the **Color links based on data from** and **using the** drop-down list boxes in the Color Links by Load dialog box) and throughput statistics respectively.

3 Click **Apply** to visualize the loads and keep the Color Links by Load dialog box open, or click **OK** to visualize the loads and close the dialog box. Click **Clear** to remove all link load visualizations from the workspace.
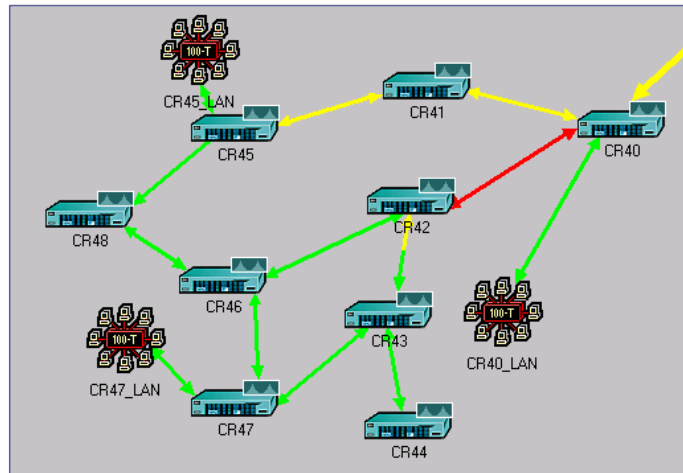
Figure 4-14   Link Loads Visualized in the Workspace

4 Move your mouse over any link to display link statistics.

End of Procedure 4-7

Sizing and Scaling Icons

You can customize the appearance of objects in your workspace by scaling the icons to a larger or smaller size, and moving icon labels that overlap. NETWARS provides commands that will perform these functions automatically for you, or you can make adjustments manually to selected icons using the Scale Selected Icons dialog box.

For example, if an icon is 12 x 12 pixels at full size, you can scale the icon to 50% so that it appears at half-size (6 x 6 pixels). If you scale an icon below a certain size, the icon becomes minimized—that is, it appears as a simple shape rather than as a bitmap.

Procedure 4-8   Automatically Scale Icons and/or Adjust Label Placement

The example provided below shows icons fixed at full size, before scaling.
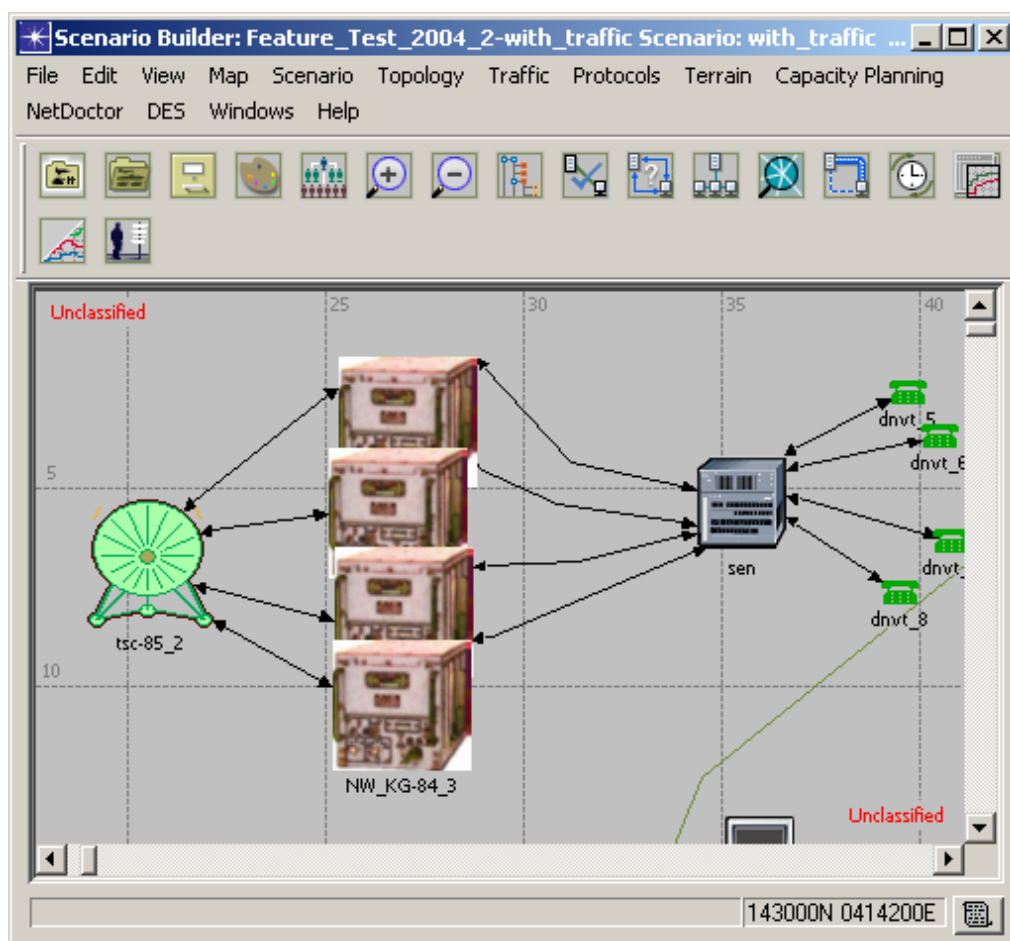
Figure 4-15   Icons Fixed at Full Size, Before Scaling

1 Select **View > Layout > Automatic Icon Scaling**. Icon sizes are adjusted automatically to minimize overlap.
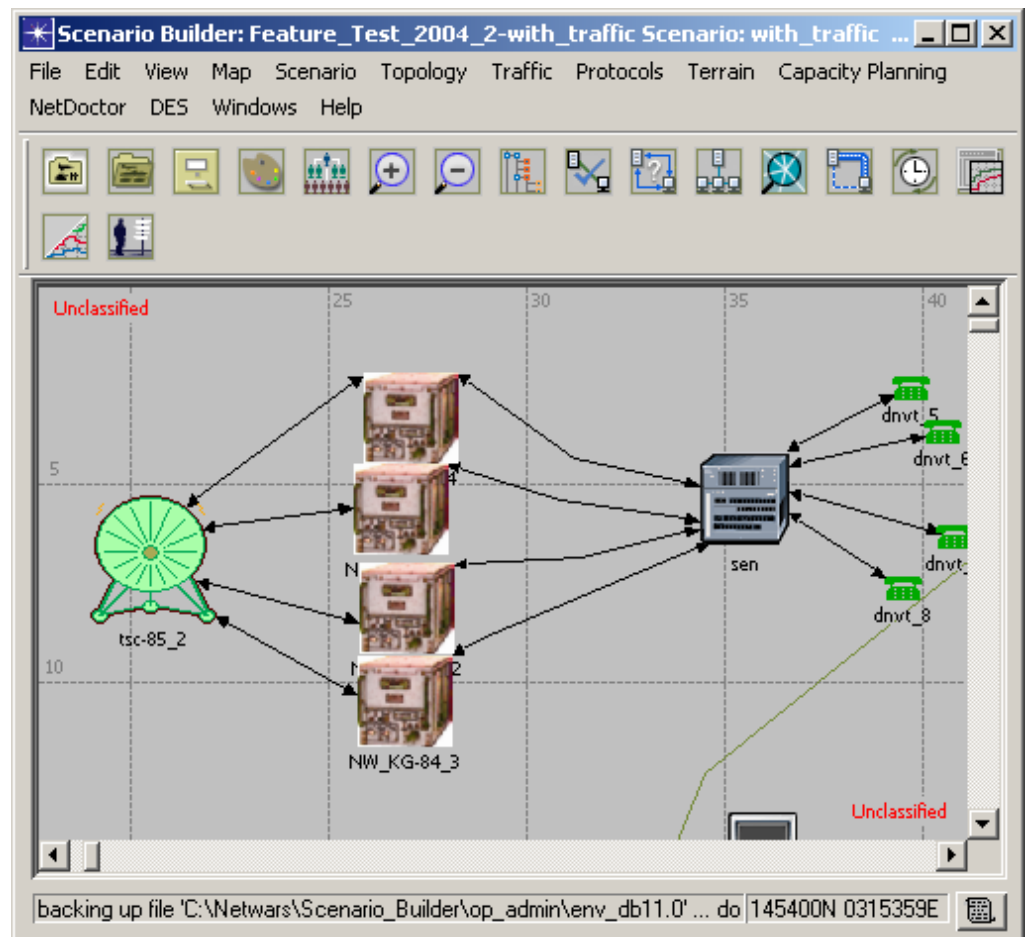

Figure 4-16   Automatically Scale Icons

2 Select **View > Layout > Automatic Label Placement**. Label positions are adjusted automatically to prevent overlap with adjacent icons.
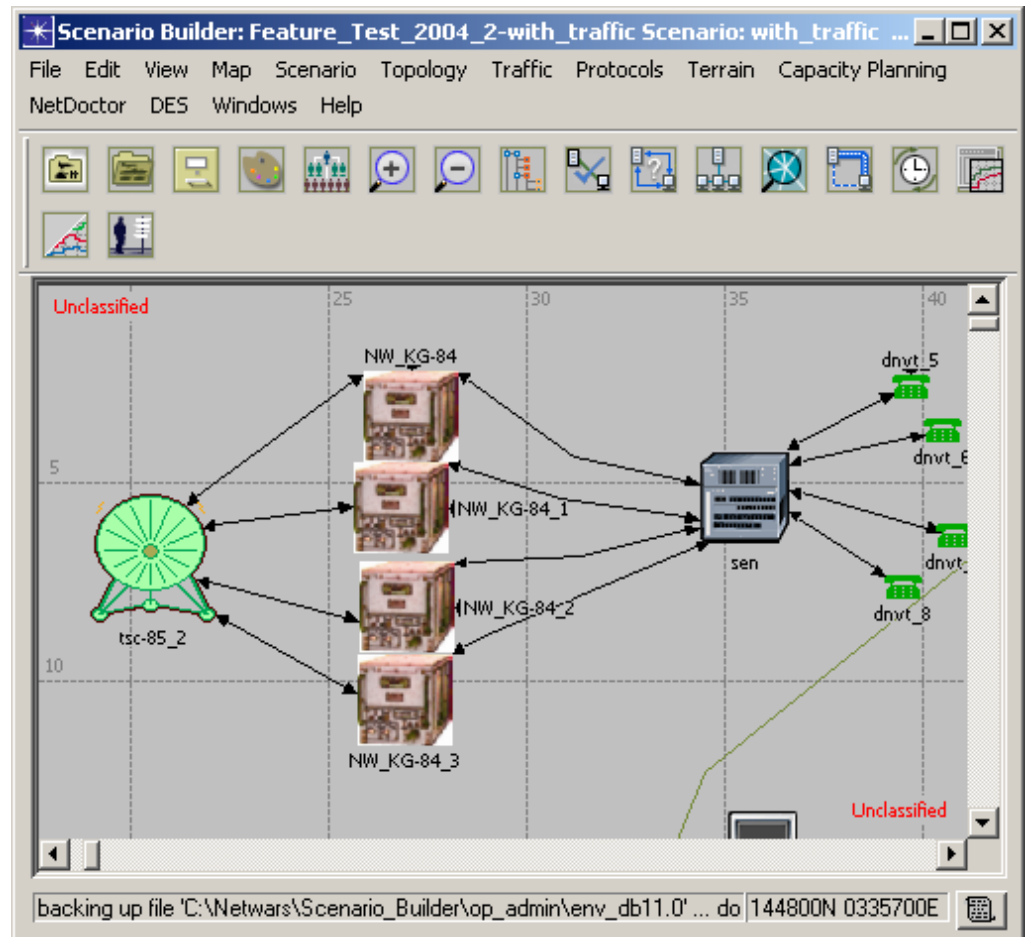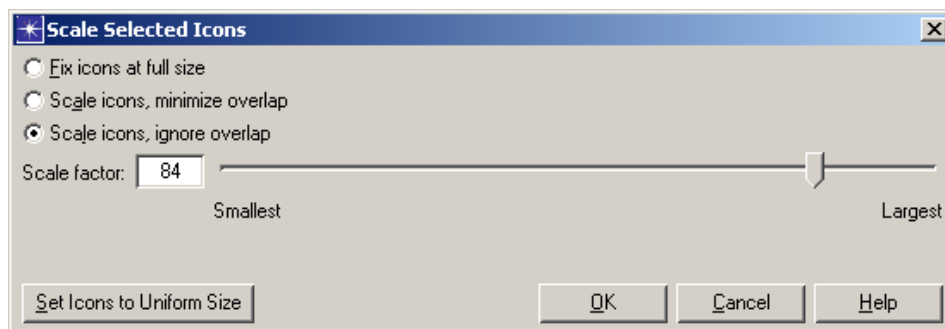

Figure 4-17   Automatically Adjust Label Placement

End of Procedure 4-8

Procedure 4-9   Manually Scale Selected Icons

1 Select the nodes whose icons you want to scale. To scale all node icons in a scenario, left-click in the network background to ensure that no nodes are selected.

2 Choose **View > Layout> Scale Selected Icons**. The Scale Selected Icons dialog box displays.

Figure 4-18   Scale Selected Icons dialog box

3 Scale icons by the desired amount:

3.1 Select the **Fix icons at full size** radio button to scale selected icons to their full size (as defined by the icon bitmaps). When zooming in or out in the workspace, the icons remain at full size regardless of the zoom level. This behavior is achieved by setting the "threshold" attribute on the selected objects to a value of 0. Selecting this option disables the icon scaling slider and the uniform size button.

3.2 Select the **Scale icons, minimize overlap** radio button to use the Scale Factor slider and field to scale all of the selected icons, except those that would overlap other icons if they were scaled any larger. The size can range from 1% to the percentage at which all of the icons will appear full size. This maximum percentage will vary, depending on the icon size when you open this dialog box.

3.3 Select the **Scale icons, ignore overlap** radio button to use the Scale Factor slider and field to scale all of the selected icons equally, even those that overlap. The size can range from 1% to the percentage at which all of the icons will appear full size. This maximum percentage will vary, depending on the icon size when you open this dialog box.

3.4 To scale all selected icons to the same size (that is, to the current size of the smallest icon in the selection set), click the **Set Icons to Uniform Size** button and then use the slider. A progress bar appears as the icons are updated.
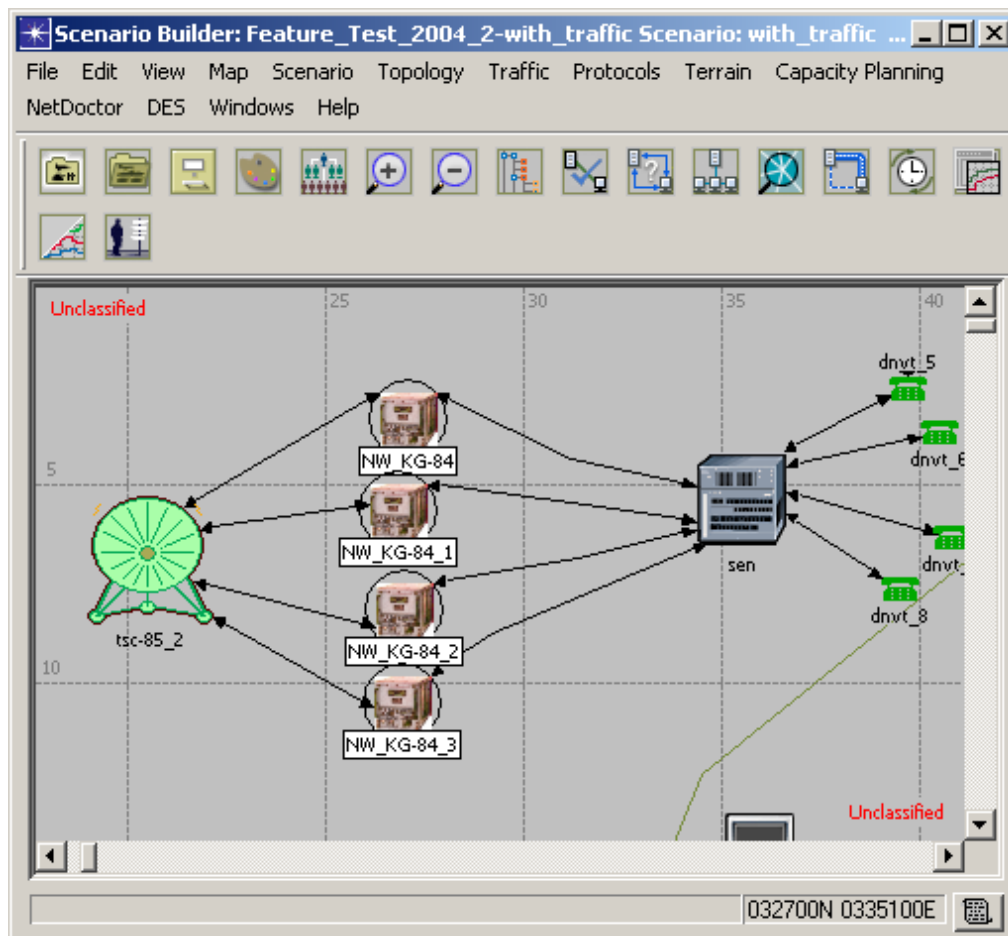

Figure 4-19   Selected Icons Scaled to a Factor of 50

3.5 Click **OK** to change the selected icons as specified and close the dialog box. A progress bar appears as the icons are updated. If no changes are necessary, the dialog box will simply close.

3.6 Click **Cancel** to close the dialog box without making any of the specified changes.

End of Procedure 4-9

Setting Visualization Preferences

You can customize the appearance of objects by editing preferences that control the visualization of the network. The preferences can be controlled on either the global or subnet level. Those preferences specified at the subnet level take precedence over any global settings. The subnet preferences are retained per subnet, and can only be changed from the Edit Visualization Preferences dialog box for the active subnet.

You can specify visualization settings (such as icon sizes to minimize, and which object labels to display and when) for nodes, links, demands, and paths.

Procedure 4-10   Set Visualization Preferences

1 Select **View > Edit Visualization Preferences**. The Edit Visualization Preferences dialog box displays.
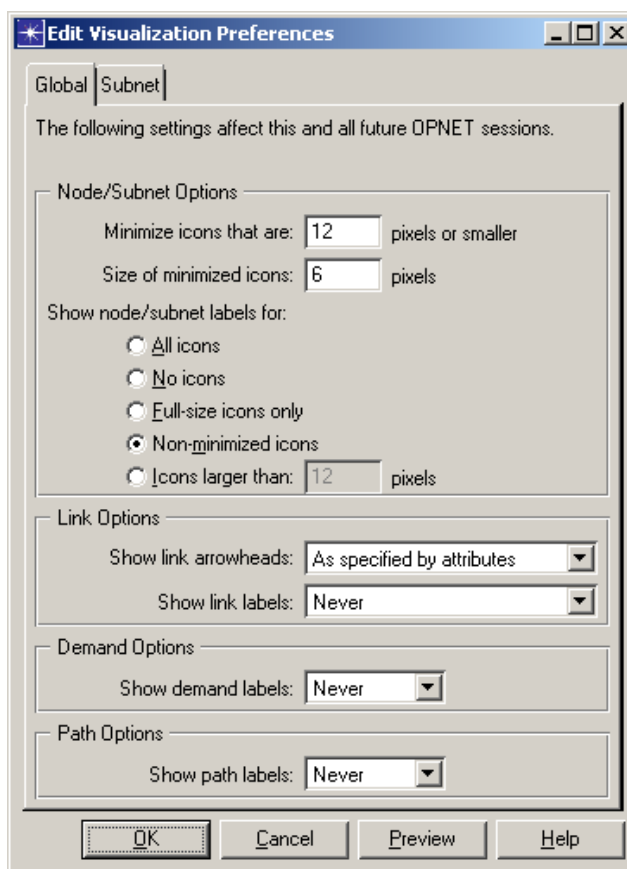
Figure 4-20   Edit Visualization Preferences dialog box

2 Type the size (in number of pixels) at which you want icons to become minimized in the **Minimize icons that are:** edit field. If an object appears smaller than the specified number of pixels, it is drawn using its minimized icon, as specified by its **minimized icon** attribute.

3 Type the size (in number of pixels) of all minimized icons in the network in the **Size of minimized icons:** edit field.

4 Click the **Show node/subnet labels for** radio button that corresponds to which object labels you want to display:

4.1 **All icons**—Labels appear on all objects.

4.2 **No icons**—No labels appear on any node or subnet objects.

4.3 **Full-size icons only**—Labels only appear on objects that have not been scaled below their original full size.

4.4 **Non-minimized icons**—Labels appear on any icons that have not been minimized.

4.5 **Icons larger than X pixels**—Labels appear on any icons that are larger than the specified number of pixels.

5 Select which links have arrowheads from the **Show link arrowheads** drop-down list:

5.1 **Never**—No arrowheads appear on any link objects.

5.2 **As specified by attributes**—The appearance of arrowheads is determined by each link object's **arrowheads** attribute.

5.3 **For simplex links only**—For all simplex links, the appearance of arrowheads is determined by each link object's **arrowheads** attribute. For all other link types, no arrowheads display.

6 Select which links have labels from the **Show link labels** drop-down list:

6.1 **Never**—No link labels appear on any link objects.

6.2 **Always**—Link labels appear on all link objects.

6.3 **For off-page links only**—Link labels appear on links that are drawn out of the active subnet.

7 Select which demands have labels from the **Show demand labels** drop-down list:

7.1 **Never**—No demand labels appear on any demand objects.

7.2 **Always**—Demand labels appear on all demand objects.

8 Select which paths have labels from the **Show path labels** drop-down list:

8.1 **Never**—No path labels appear on any path objects.

8.2 **Always**—Path labels appear on all path objects.

9 Click **OK** to close the dialog box. All settings take effect immediately, including modification of global environment preferences and subnet-level settings.

Or

Click **Cancel** to close the dialog box and return to the original display without making changes to the visualization settings, global environment attributes, or subnet-level settings.

Or

Click **Preview** to make the changes take effect immediately; however, the environment preferences and subnet settings will not be retained unless you also click **OK**.
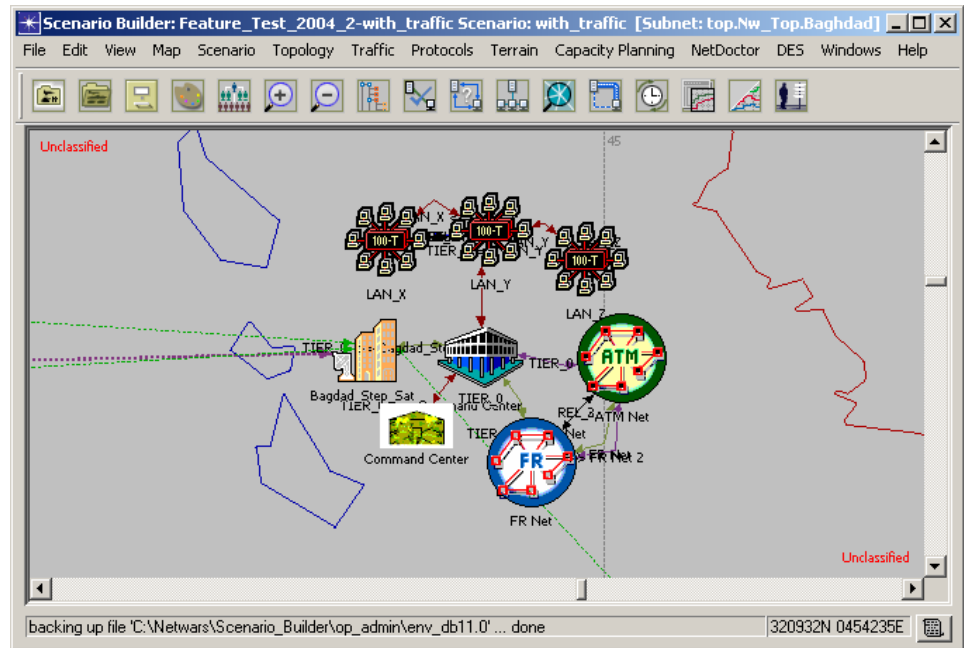

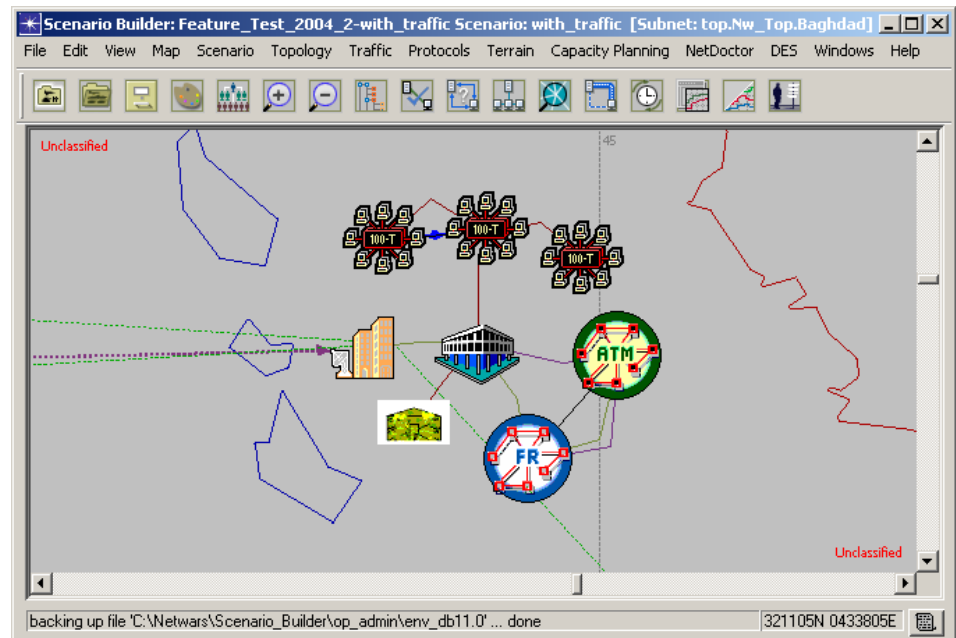**Figure 4-21   Preferences Set to Show Icon and Link Labels and Arrowheads**

Figure 4-22   Preferences Set to Not Show Icon and Link Labels or Arrowheads

End of Procedure 4-10

Using the Time Controller

NETWARS can model many types of data that vary over time, such as link loads or utilizations, traffic levels between two nodes, simulation results that vary over time, and the locations of mobile nodes in wireless scenarios. NETWARS can measure and represent data using calendar times (for example, "Jan 12 19:40:30 2005") or relative times, which are offsets from a start time (for example, "30 seconds"). A scenario can have a network start time associated with it. The start time is a calendar time (that is, an absolute date/time) value that defines the start of the time window for which the scenario models the network. When you run a discrete event simulation, for example, NETWARS does the following:

- Treats all relative time values as offsets from the network start time, and

- Ignores any time-based data outside the simulation time window, which starts at *<network_start_time>* and ends at *<network_start_time>* + *<simulation_duration>*.

To see how time-varying data changes over time, use the Time Controller. It allows you to change the current network time visualized in the editor and animate the time change effect.

Procedure 4-11   Use the Time Controller

1   Select **View > Show Time Controller** (or press **<Ctrl>**+**<Shift>**+**T**). The Time Controller displays.
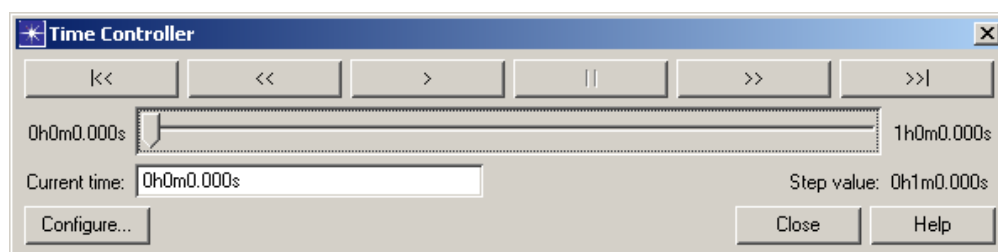
Figure 4-23   Open the Time Controller

The Time Controller shows the current network time (text field); the start time, stop time, and step values (read-only text fields); buttons to change time by stepping through the time steps or through an animation; and a time slider which provides an alternative way to change the current time.

2   Click **|<< (Rewind)** to reset network time to slider start time.

3   Click **<< (Step back)** to decrease network time by one time step.

4   Click **> (Play)** to start animation.

5   Click **|| (Pause)** to stop animation.

6   Click **>> (Step forward)** to advance network time by one time step.

7 Click **>>| (Fast forward)** to set network time to slider end time.

8 Click and slide the time slider bar if you'd like an alternative way to change the current network time.

9 Click **Configure** if you want to change Time Controller settings. The Time Controller Settings dialog box displays.
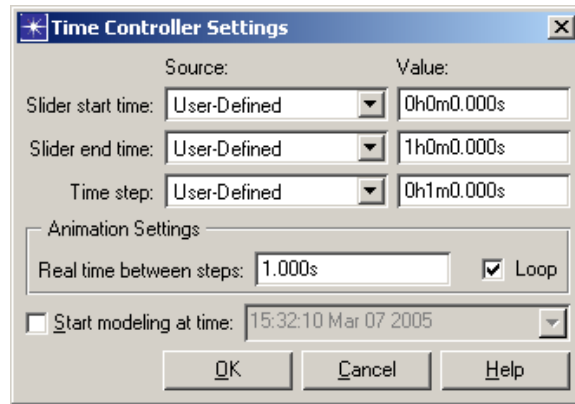

Figure 4-24   Time Controller Settings dialog box

9.1 Select the source plugin for the slider start time value from the **Slider start time:** drop-down list. If 'User-Defined' is selected, you can enter the value in the corresponding **Slider start time:** text field. The value should be in ##h##m##.###s format.

9.2 Select the source plugin for the slider end time value from the **Slider end time:** drop-down list. If 'User-Defined' is selected, you can enter the value in the corresponding **Slider end time:** text field. The value should be in ##h##m##.###s format.

9.3 Select the source plugin for the time step value from the **Time step:** drop-down list. If 'User-Defined' is selected, you can enter the time step in the corresponding **Time step:** text field. The value should be in ##h##m##.###s format.

9.4 Type a time (in number of seconds) in the **Real time between steps:** field to control the speed of the animation.

9.5 Check the **Loop** checkbox to restart animation after completion.

9.6 Check the **Start modeling at time:** checkbox if you want to specify a calendar time for the network start time in the **Start modeling at time:** drop-down list box. The value should follow "hours:minutes:seconds month date year" format. Otherwise, the start time is assigned a symbolic value of "time 0.0".

9.7 Click **OK** to accept changes and close the dialog box.

10 Click **Close** to close the Time Controller.

End of Procedure 4-11

5   Reports

NETWARS provides several reporting features that make it easier and faster for you to create and review reports, tables, and graphs to help you to understand your network configuration. This chapter discusses the following types of reports:

- **Summary Tables**—View summary information for a scenario in table format.

- **User-defined Report Tables**—Get a quick configuration overview of specific attributes of a number of network objects, and generate publishable HTML reports.

- **Capacity Planning Web Reports**—Review the current state of a scenario after performing a capacity planning evaluation. The web reports launch automatically once the evaluation is complete.

Using Summary Tables

NETWARS provides immediate access to summary information for a scenario in table format. Available tables include a summary of: Free-Text Questions, IP Addresses, Links, Mission Analysis, Requirements, and SLDs. NETWARS allows you to print summary table data, or export the information displayed in a table to a spreadsheet (.xls) or HTML file.

Procedure 5-1   Open a Summary Table

1   Select **Scenario > Summary Tables > Free-Text Questions**. The Summary of Free-text Questions dialog box displays a summary of free-text questions attached to organizations, OPFACs, and devices.
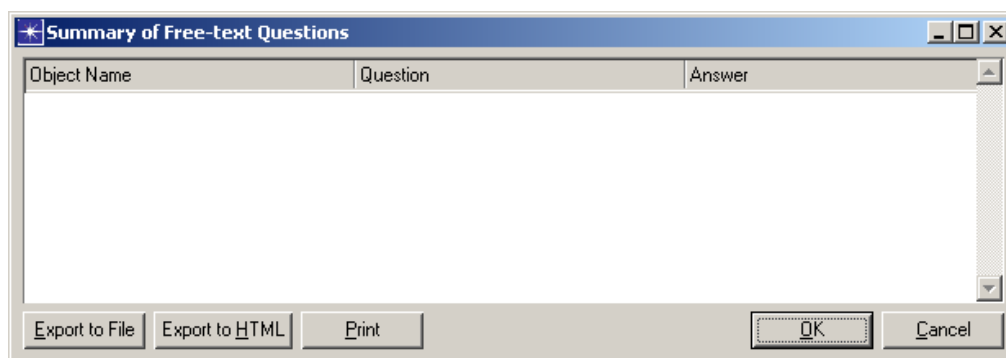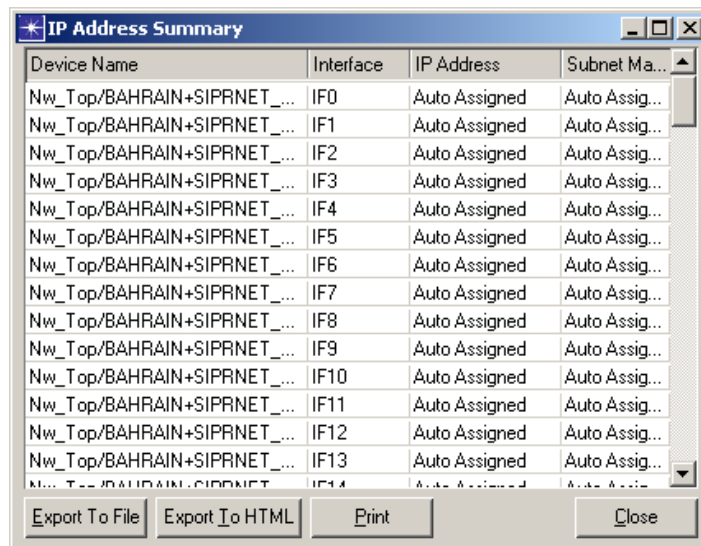
Figure 5-1   Summary of Free-text Questions dialog box

The Summary of Free-text Questions dialog box provides a mechanism for the planners to communicate with each other using free text. This is especially useful when a Subordinate Planner wants to communicate with a Lead Planner since the mission analysis question interface cannot import back new questions added by the Subordinate Planner. These questions and answers are stored as attributes on the objects and can be accessed by right-clicking on the object and selecting **Edit Attributes** and clicking the Value column next to the Free-Text Questions attribute.

Note—You cannot modify the values displayed in this table.

1.1 Click **Export to File** to export table data to a spreadsheet (.xls) file.

1.2 Click **Export to HTML** to export table data to an HTML file.

1.3 Click **Print** to print the table to the selected printer.

1.4 Click **Close** to close the Summary of Free-text Questions dialog box.

2 Select **Scenario > Summary Tables > IP Addresses**. The IP Address Summary dialog box displays IP addresses for all connected interfaces on the devices.

Figure 5-2   IP Address Summary dialog box

Note—You cannot modify the values displayed in this table.

2.1 Click **Export to File** to export table data to a spreadsheet (.xls) file.

2.2 Click **Export to HTML** to export table data to an HTML file.

2.3 Click **Print** to print the table to the selected printer.

2.4 Click **Close** to close the IP Address Summary dialog box.

3 Select **Scenario > Summary Tables > Links**. The Link Summary dialog box displays a list of all links in the scenario along with some key attributes.

Figure 5-3   Link Summary dialog box

Note—You cannot modify the values displayed in this table.

3.1 Click **Export to File** to export table data to a spreadsheet (.xls) file.

3.2 Click **Export to HTML** to export table data to an HTML file.

3.3 Click **Print** to print the table to the selected printer.

3.4 Click **Close** to close the Links Summary dialog box.

4 Select **Scenario > Summary Tables > Mission Analysis**. The Mission Analysis Questions dialog box displays the default mission analysis questions that are applied to all new projects created in the Scenario Builder.

Figure 5-4   Mission Analysis Questions dialog box

4.1 Click **Situation**, **Mission**, **Execution**, **Admin & Logistics**, or **Command & Control** to filter the displayed table results by functional area.

4.2 Click **Add Question** to add a new question or **Delete Question** to remove an existing question from the list.

4.3 Click **Export to File** to export table data to a spreadsheet (.xls) file.

4.4 Click **Export to HTML** to export table data to an HTML file.

4.5 Click **Print** to print the table to the selected printer.

4.6 Click **Import** to import additional mission analysis questions in to the Mission Analysis table instead of manually typing them. The text file must be tab-delimited and follow the same format as the Mission Analysis defaults file, i.e., MissionAnalysis.gdf.

4.7 Click **OK** to save any changes and close the Mission Analysis Questions dialog box, or **Cancel** to close the dialog box without saving.

5 Select **Scenario > Summary Tables > Requirements**. The Requirements Entry Summary dialog box displays a summary of the bandwidth requirements for all the organizations in the scenario.

Figure 5-5   Requirements Entry Summary dialog box

The table displays one row for every top-level organization in the scenario. Each row has the name of the organization followed by the required bandwidth for each service type. The default functional area is Voice.

5.1 Click **Voice Services**, **Data Services**, **VTC Services**, **Message Services**, or **Joint Digital Network** to filter the displayed table results by functional area.

5.2 Click **Export to File** to export table data to a spreadsheet (.xls) file.

5.3 Click **Export to HTML** to export table data to an HTML file.

5.4 Click **Print** to print the table to the selected printer.

5.5 Click **Close** to close the Requirements Entry Summary dialog box.

6 Select **Scenario > Summary Tables > SLDs**. The SLD Summary dialog box displays all the SLDs specified in the scenario. The table lists all external, radio, and satellite links regardless of whether or not they use the SLD naming scheme.

Figure 5-6   SLD Summary dialog box

A SLD is a System Link Designator that connects two transmission systems/ devices. It follows an eight-character alphanumeric code to identify the transmission systems it connects. In NETWARS, transmission systems are connected by satellite, external, radio and internal links as well as broadcast networks.

The SLD Name column includes the SLD names or the previous device_a <-> device_b / sat_link_<n> names. The From "A" Loc and To "B" Loc columns include the corresponding From and To codes. The "A" Equip and "B" Equip columns are based on the self-description for the A and B devices. If no self-description exists for these models, names are used that are based on the model attribute on the device. The Responsible Unit column values directly correspond to the RCU values specified in the JNMS Attributes dialog box for the device's parent organization. The From "A" Device and To "B" Device columns provide the hierarchical name of the end devices.

Note—You cannot modify the values displayed in this table.

6.1 Select a name in the SLD Name column to zoom in to the corresponding link in the workspace.

6.2 Click **Export to File** to export table data to a spreadsheet (.xls) file.

6.3 Click **Export to HTML** to export table data to an HTML file.

6.4 Click **Print** to print the table to the selected printer.

6.5 Click **Close** to close the SLD Summary dialog box.

End of Procedure 5-1

Creating User-Defined Reports

NETWARS allows you to create user-defined report tables that enable you to get a quick configuration overview of specific attributes of a number of network objects, and generate publishable HTML reports.

Procedure 5-2   Create User-Defined Reports

1   Select **Scenario > Summary Tables > User-Defined Reports > Generate Report from Template...**. The Generate User-Defined Report dialog box displays.
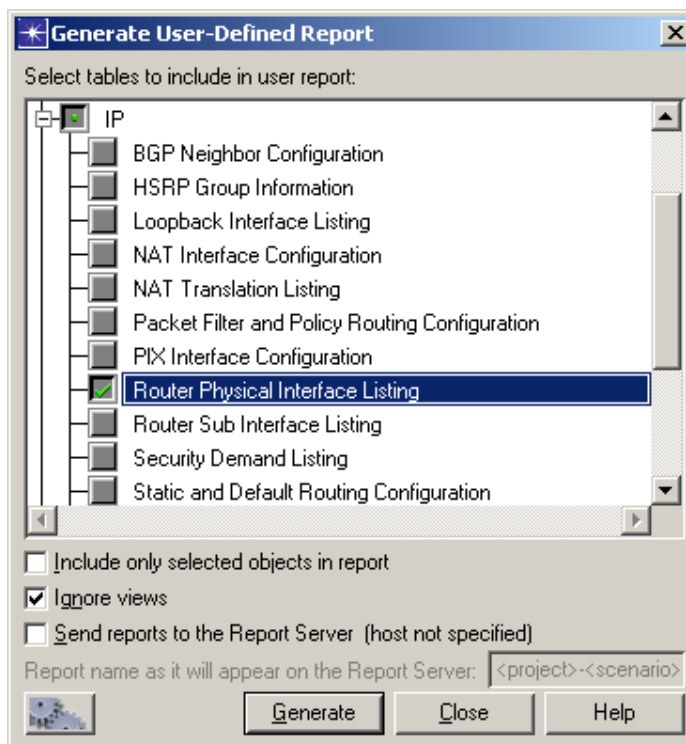
Figure 5-7   Generate User-Defined Report dialog box

2   Select the tables to include in the report.

To select an entire category of tables, click on the top-level category entries. To select an individual table, expand the top-level category subtree and click on the table entry. Selected tables and categories show a green checkmark. Categories with some but not all of its tables selected will display a green dot instead.

3   Select the **Include only selected objects in report** checkbox if you want to collect table data only for objects that are currently selected in the network.

4   Select the **Ignore views** checkbox if you want to collect table data even for objects that are not in the current view or views of the network.

5   Select the **Send reports to the Report Server (*host*)** checkbox if you want to send reports to the Report Server. This option is active only when a Report Server license is available.

5.1   Specify a name for the report in the Report Server text field.

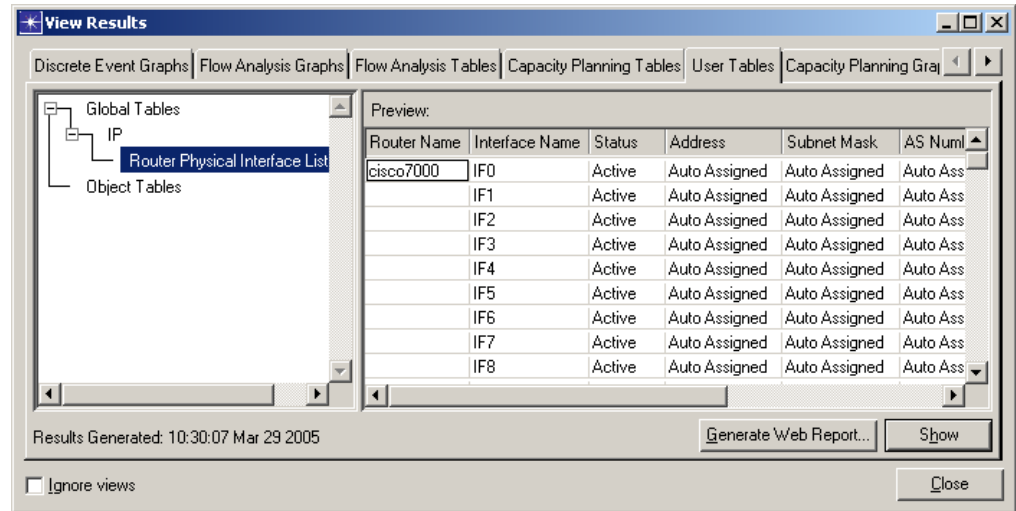**6** Click **Generate**. The View Results dialog box displays.

Figure 5-8   View Results dialog box

Note—You can also right-click an object in your network to display the shortcut menu, and select the **View Results** command to launch the View Results dialog box with only the graphs and tables of that object.
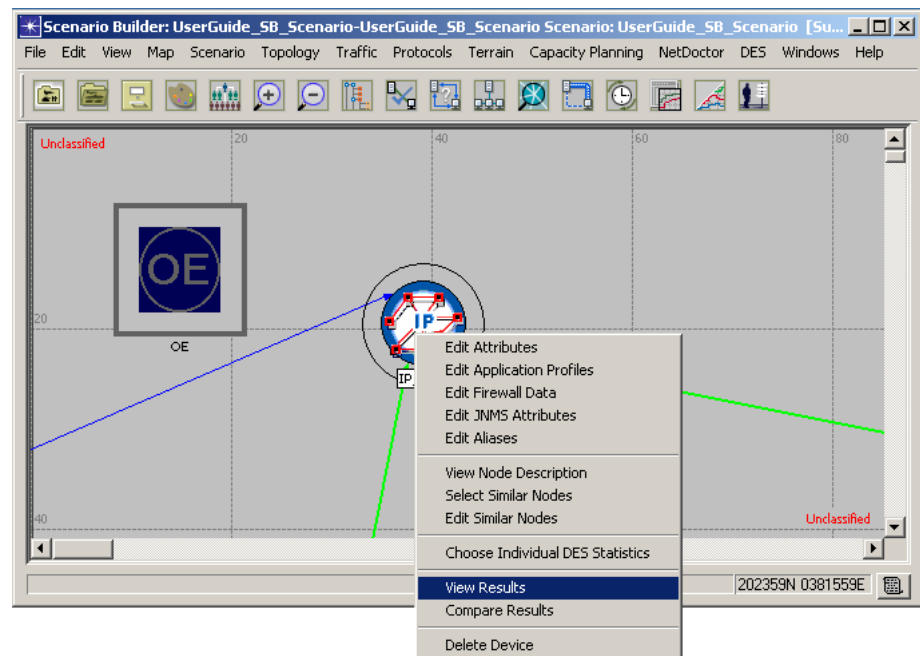

Figure 5-9   View Results via the Shortcut Menu

The left side of the View Results dialog box shows a result hierarchy, and the right side shows a rough preview (either in table or graph format.)

Several tabs are displayed above the results hierarchy and preview windows in the View Results dialog box.

7 Click on a tab in the View Results dialog box to bring it forward (making it active) and preview any associated table or graph data. You can choose from the following options:

- Discrete Event Graphs tab (also displayed by selecting **DES > Results > View Results**)
- Flow Analysis Graphs tab
- Flow Analysis Tables tab
- Capacity Planning Tables tab
- User Tables tab
- Capacity Planning Graphs tab
- Displayed Panel Graphs tab

8 Click **Show** in the View Results dialog box to display a preview of the table or graph data for the active tab.

Figure 5-10   Table Data Preview

8.1 Use the graph options/filters drop-down menus to display graphs in a variety of layouts.

8.2 Right-click in the graph area to open a shortcut menu that allows manipulation of graph properties. Right-click on the margin of a graph to open a shortcut menu with different graph-manipulation options.
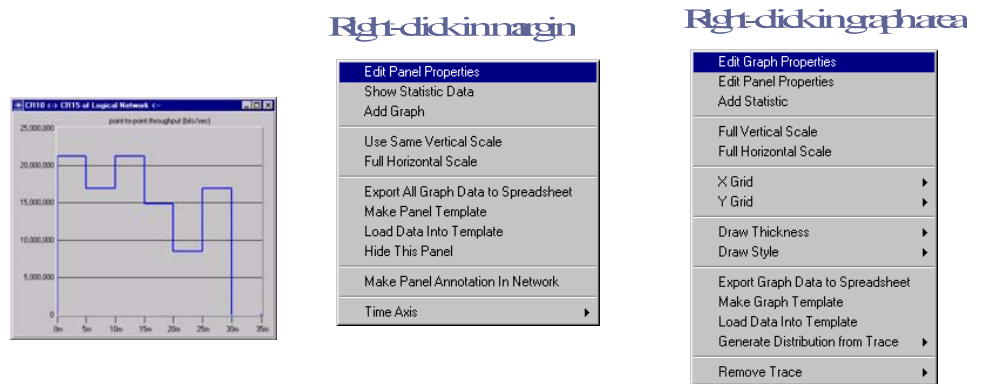
Figure 5-11   Graph Data Preview

9 Click **Generate Web Report** in the View Results dialog box to generate an HTML-based web report. The Select Tables for Web Report dialog box displays.
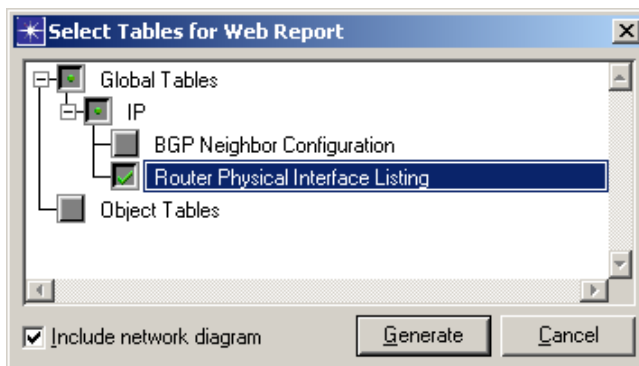
Figure 5-12   Select Tables for Web Report dialog box

9.1 From the Select Tables for Web Report dialog box, choose the results you want to include in the Web report.

9.2 Check the **Include network diagram** checkbox if you want the web report to include a graphic of the network topology.

9.3 Click **Generate**.The Browse for Folder dialog box displays.

Figure 5-13   Browse for Folder dialog box

9.4 Select a directory for the HTML files and click **OK**. The web report is created and opens in the default Web browser.
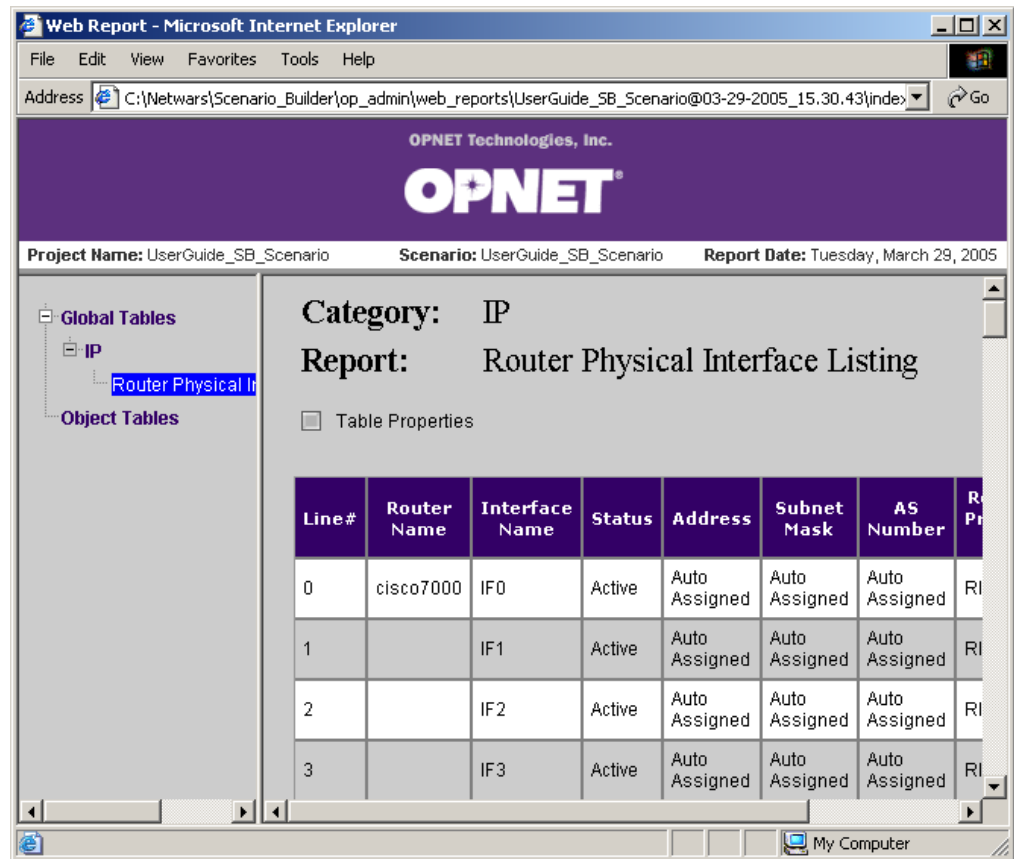
Figure 5-14   Sample Web Report

End of Procedure 5-2

Opening User-Defined Report Tables in Live Mode

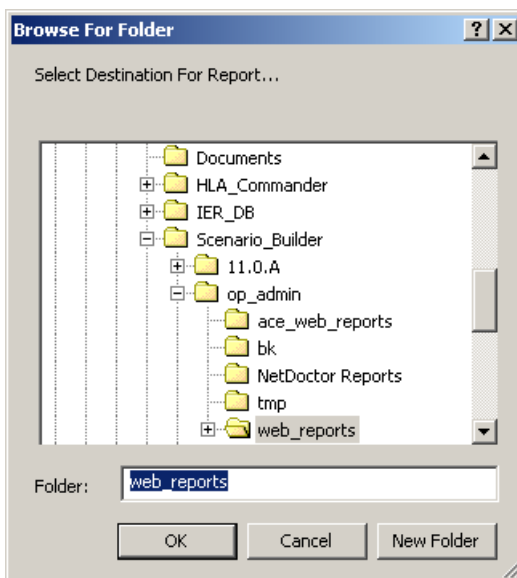NETWARS allows you to open one or more user-defined report tables in *live* mode. This means the tables will not only show the corresponding values given the current selection set, but will also automatically repopulate themselves whenever the selection set changes. This provides a quick way of surveying the relevant attributes of a number of network objects without generating a full report and searching for the rows of interest. It can also sometimes be more handy than, for example, opening the Edit Attributes dialog box for each of the objects of interest.

Note—This functionality is supported independently of the **Generate Report from Template** functionality, so it will neither create a new report nor discard any existing report information.

Procedure 5-3   Open User-Defined Report in Live Mode

1 Select **Scenario > Summary Tables > User-Defined Reports > Open Live Report Table...**. The Live User-Defined Report Tables dialog box displays.
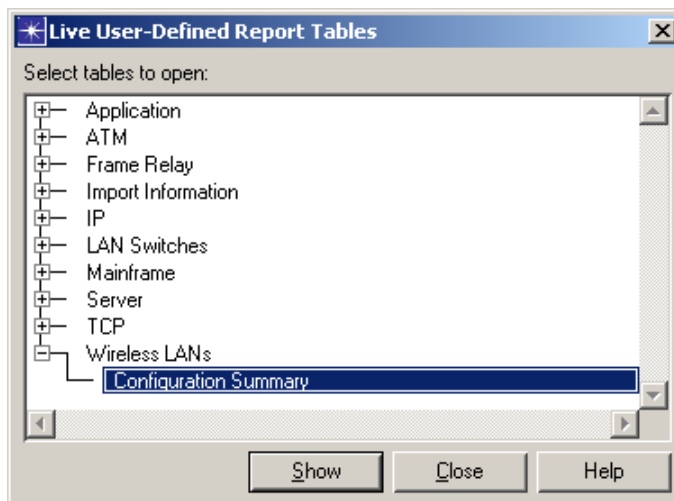
Figure 5-15   Live User-Defined Report Tables dialog box

2 Specify which table of which category should be opened in the Select tables to open treeview. Expand the top-level category subtree and click on the desired table entry. You can select multiple tables by holding down the **<Ctrl>** key while left-clicking or by click-and-dragging a rectangular selection. Selecting a top-level category tree row itself does not imply selecting all or any of its tables (it is simply ignored).

3 Click **Show** to build and open individual user-defined report table dialog boxes for each of the tables you have selected.

4 Click **Close** to close the dialog box.

End of Procedure 5-3

Using Capacity Planning Web Reports

After performing a capacity planning evaluation, you can use the resulting web reports to review the current state of a scenario. The web reports launch automatically once the evaluation is complete. Subsequently, you can open the web reports by selecting them individually by name from the **Capacity Planning > Reports** menu where they are listed. You can also open the most recent set of Capacity Planner web reports for the current scenario by selecting **Capacity Planning > Reports > View Web Report**.

The web reports include the following reports for the overall evaluation:

- Executive Summary
- Overall Peak Results
- Overall Average Results

Additionally there are a group of reports which are generated for each time step:

- **Traffic Report**—The Traffic Route Report contains the Traffic ID, the source of the traffic (IER, Flow or Application), the traffic type (voice or data), the priority, the load in kbps, a link to a secondary table giving the routing information, and the source and destination of the traffic. Additionally, the source and destination are hyperlinked to the source and destination nodes in the workspace. Unroutable demand entries are highlighted in red and link to an Unroutable Traffic report, which contains the reason the traffic could not be routed and possible debugging suggestions. The sorting in the table is done first over unroutable versus routable demands with unroutable demands positioned higher, then the table is sorted by traffic source, followed by destination and finally load, with higher loads located higher.

- **Link Utilization Report**—The Link Utilization Report contains the link name, its data rate in kbps, the maximum total utilization, the total forward utilization, the total reverse utilization, the forward data utilization, the reverse data utilization, the forward circuit utilization, the reverse circuit utilization, and the voice utilization. Under the link name, there are hyperlinks to both the link object in the workspace and to a report containing that specific link's traffic and circuits. The table is sorted by the max total utilization. Links that have very high max total utilizations are highlighted in red and links that have somewhat high max total utilizations are highlighted in yellow. The default values for highlighting are 75% and 60% respectively. You can select the value for which links with greater utilization have highlighted entries.

- **Circuit Reports (Promina, CellXpress & ATM, and FCC-100)**—Circuit Reports contain the names of the source and destination device as well as the names and ports of the end source and destination circuits. All of these table entries are hyperlinked to the nodes in the workspace. Additionally, the report contains the data rate of the circuit in kbps, the direction of the circuit (forward, duplex or reverse) and a hyperlink to the route table of the circuit. Table entries are highlighted in red if they were unroutable and contain a link to an Unroutable Circuit Report, which gives a reason why the circuit was

unroutable and debugging information. The report is sorted first by routable, then unroutable, with unroutable routes on top and highlighted in red. Further sorting is done by source name, and then further sorted by destination and finally by data rate.

There are also Route Reports for each routed circuit and traffic demand, and Unroutable Reports for each unroutable circuit and traffic demand. Each Individual Link also has a group of tables that show the circuit and traffic demands which flow over it specifically.

- **Route Table**—The Route Table lists each hop the traffic takes, including the source, destination, and link that is traversed. The source and destination are both hyperlinked to the object nodes in the workspace. The link also has hyperlinks to the link object in the workspace and the Individual Link Traffic Report for the link.

- **Unroutable Circuit Report**—The Unroutable Circuit Report contains the source and destination devices for the circuit, which are hyperlinked to the workspace. The report also contains a reason for why the circuit was unroutable and debugging suggestions.

- **Unroutable Traffic Report**—The Unroutable Traffic Report contains the source and destination devices with hyperlinks to the workspace. The report also contains a reason for why the traffic was unroutable and debugging suggestions.

- **Individual Link Traffic Report**—The Individual Link Traffic Report contains three tables: forward traffic over the link, reverse traffic over the link, and circuits over the link. Depending upon the link, some of these tables may be empty. The format of the traffic tables is the same as in the Traffic Route report. The format of the circuit table is the same as the Promina Circuit Route Report.

Each report contains hyperlinks explaining how the statistics were calculated. Each report also includes the date, project and scenario name in its header. Due to the long hierarchical names sometimes used in models, you can view the reports with variable name lengths. The default value is opfac.device. You can change this setting before running an evaluation as discussed in Procedure 4-4.

Procedure 5-4   Set Capacity Planning Web Report Parameters

1   Select **Capacity Planning > Reports > Settings**. The Web Report Settings dialog box displays.
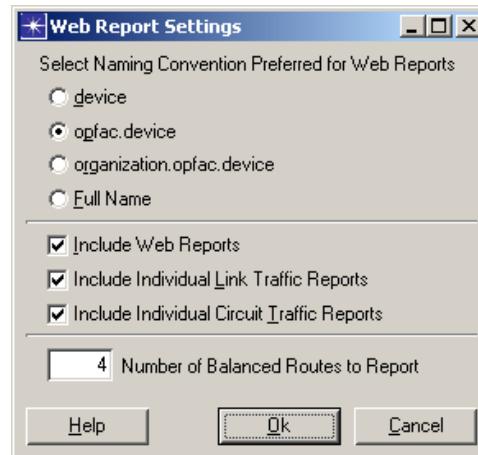
Figure 5-16   Web Report Settings dialog box

2   In the Select Naming Convention Preferred for Web Reports section, click the radio button that corresponds to the naming convention you want to use for all the devices and links that are in the report.

3   Check the **Include Web Reports** checkbox if you also want to select the link and circuit traffic reports. Unchecking this option will disable the options for the individual reports.

4   Check the **Include Individual Link Traffic Reports** checkbox if you want individual reports to be included in the web report for each link in the scenario. The reports will include all the traffic going directly over the link as well as any circuits that are going over the link. The reports will take longer to generate if this option is selected.

5   Check the **Include Individual Circuit Traffic Reports** checkbox if you want individual reports to be included in the web report for each routable circuit. The reports will include all the traffic using the circuit. The reports may take longer to generate if this option is selected.

6   In the Number of Balanced Routes to Report field, type the number of balanced routes that you want reported in the web reports. The entry must be an integer greater than 0.

7   Click **OK** to save your settings, or **Cancel** to close the Web Report Settings dialog box without saving your changes.

End of Procedure 5-4

Procedure 5-5   Open Capacity Planning Web Reports

1   Select **Capacity Planning > Reports > *<report name>***. The individually selected Capacity Planning Report for the current scenario displays in a web browser window.

2   Select **Capacity Planning > Reports > View Web Report**. The most recent set of Capacity Planning Reports for the current scenario displays in a web browser window.
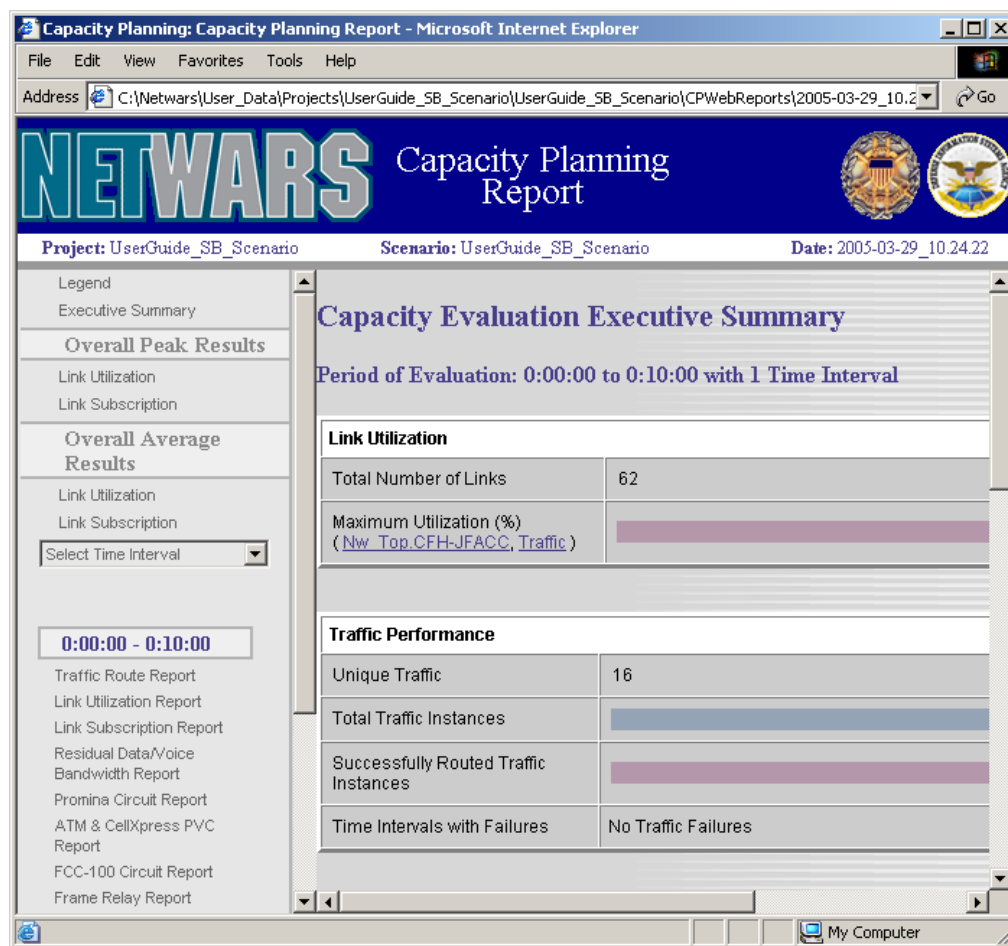
Figure 5-17   Capacity Planning Web Report

The report opens to the Executive Summary, which gives you a short overview of the report highlights. More detailed results are available by clicking on one of the links provided on the left-hand side of the window.

End of Procedure 5-5