



Cisco Application Analysis Solution General Tutorials

Release 1.0

Corporate Headquarters

Cisco Systems, Inc.
170 West Tasman Drive
San Jose, CA 95134-1706
USA
<http://www.cisco.com>
Tel: 408 526-4000
800 553-NETS (6387)
Fax: 408 526-4100

Text Part Number: OL-7527-01



THE SPECIFICATIONS AND INFORMATION REGARDING THE PRODUCTS IN THIS MANUAL ARE SUBJECT TO CHANGE WITHOUT NOTICE. ALL STATEMENTS, INFORMATION, AND RECOMMENDATIONS IN THIS MANUAL ARE BELIEVED TO BE ACCURATE BUT ARE PRESENTED WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED. USERS MUST TAKE FULL RESPONSIBILITY FOR THEIR APPLICATION OF ANY PRODUCTS.

THE SOFTWARE LICENSE AND LIMITED WARRANTY FOR THE ACCOMPANYING PRODUCT ARE SET FORTH IN THE INFORMATION PACKET THAT SHIPPED WITH THE PRODUCT AND ARE INCORPORATED HEREIN BY THIS REFERENCE. IF YOU ARE UNABLE TO LOCATE THE SOFTWARE LICENSE OR LIMITED WARRANTY, CONTACT YOUR CISCO REPRESENTATIVE FOR A COPY.

The Cisco implementation of TCP header compression is an adaptation of a program developed by the University of California, Berkeley (UCB) as part of UCB's public domain version of the UNIX operating system. All rights reserved. Copyright © 1981, Regents of the University of California.

NOTWITHSTANDING ANY OTHER WARRANTY HEREIN, ALL DOCUMENT FILES AND SOFTWARE OF THESE SUPPLIERS ARE PROVIDED "AS IS" WITH ALL FAULTS. CISCO AND THE ABOVE-NAMED SUPPLIERS DISCLAIM ALL WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING, WITHOUT LIMITATION, THOSE OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OR ARISING FROM A COURSE OF DEALING, USAGE, OR TRADE PRACTICE.

IN NO EVENT SHALL CISCO OR ITS SUPPLIERS BE LIABLE FOR ANY INDIRECT, SPECIAL, CONSEQUENTIAL, OR INCIDENTAL DAMAGES, INCLUDING, WITHOUT LIMITATION, LOST PROFITS OR LOSS OR DAMAGE TO DATA ARISING OUT OF THE USE OR INABILITY TO USE THIS MANUAL, EVEN IF CISCO OR ITS SUPPLIERS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

CCSP, CCVP, the Cisco Square Bridge logo, Follow Me Browsing, and StackWise are trademarks of Cisco Systems, Inc.; Changing the Way We Work, Live, Play, and Learn, and iQuick Study are service marks of Cisco Systems, Inc.; and Access Registrar, Aironet, ASIST, BPX, Catalyst, CCDA, CCDP, CCIE, CCIP, CCNA, CCNP, Cisco, the Cisco Certified Internetwork Expert logo, Cisco IOS, Cisco Press, Cisco Systems, Cisco Systems Capital, the Cisco Systems logo, Cisco Unity, Empowering the Internet Generation, Enterprise/Solver, EtherChannel, EtherFast, EtherSwitch, Fast Step, FormShare, GigaDrive, GigaStack, HomeLink, Internet Quotient, IOS, IP/TV, iQ Expertise, the iQ logo, iQ Net Readiness Scorecard, LightStream, Linksys, MeetingPlace, MGX, the Networkers logo, Networking Academy, Network Registrar, *Packet*, PIX, Post-Routing, Pre-Routing, ProConnect, RateMUX, ScriptShare, SlideCast, SMARTnet, StrataView Plus, TeleRouter, The Fastest Way to Increase Your Internet Quotient, and TransPath are registered trademarks of Cisco Systems, Inc. and/or its affiliates in the United States and certain other countries.

All other trademarks mentioned in this document or Website are the property of their respective owners. The use of the word partner does not imply a partnership relationship between Cisco and any other company. (0502R)

General Tutorials

Copyright © 2005 Cisco Systems, Inc. All rights reserved.

Copyright

Document Copyright

Title: General Tutorials
Part Number: D00203
Version: 10

© 1987-2005 OPNET Technologies, Inc.

All Rights Reserved. Reproduction, adaptation, or translation without prior written permission is prohibited, except as allowed under the copyright laws.

Software Copyright

Product Name: IT Guru
Product Release: 11.0

© 1987-2005 OPNET Technologies, Inc.
All Rights Reserved.

Documentation Conventions

OPNET documentation uses specific formatting and typographic conventions to present the following types of information:

- Objects, examples, and system I/O
- Object hierarchies, notes, and warnings
- Computer commands
- Lists and procedures

Objects, Examples, and System I/O

- Directory paths and file names are in plain Courier typeface:

```
opnet\release\models\std\ip
```

- 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 (<>):

```
<opnet_user_home>/op_admin/err_log
```

Object Hierarchies, Notes, and Warnings

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

```
Open File > Print Setup > Properties...
```

Attribute hierarchies are represented by angled arrows (▲) that indicate that you must drill down to a lower level of the hierarchy:

Attribute level 1 ▶ Attribute level 2 ▶ Attribute level 3

Note—Notes are indicated by text with the word Note at the beginning of the paragraph. Notes advise you of important supplementary information.

WARNING—Warnings are indicated by text with the word WARNING at the beginning of the paragraph. Warnings advise you of vital information about an operation or system behavior.

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. A step may be followed by a description of the results of that step; such descriptions are preceded by an arrow.

Procedure FM-1 Sample Procedure Format

- 1 Procedure step.
 - ➔ Result of the procedure step.

- 2 Procedure step.

End of Procedure FM-1

For more information about using and maintaining OPNET documentation, see the OPNET IT Guru Documentation Guide.

Contents

Copyright TUT-FM-iii

Documentation Conventions TUT-FM-iv

About the Tutorials in This Book **TUT-INTRO-1**

1 Introduction to IT Guru **TUT-1-1**

2 Small Internetworks **TUT-2-1**

3 LAN Modeling **TUT-3-1**

4 Web Reporting **TUT-4-1**

5 Importing Traffic **TUT-5-1**

6 Expert Service Prediction **TUT-6-1**

7 Productivity Features **TUT-7-1**

App A Troubleshooting Tutorials **TUT-A-1**

About the Tutorials in This Book

The tutorials in this book are the same tutorials that are delivered on the Documentation CD that comes with the OPNET software installation package. You can access the tutorials by choosing Help > Tutorials from the main menu of your OPNET application.

The tutorials are printed here in the same format as they appear online. This ensures page-by-page equivalency with the electronic version. Consequently, any hypertext links or action buttons that help you navigate the tutorials online are not active in the printed version. However, because it is frequently referenced from many of the tutorials, App A Troubleshooting Tutorials on page TUT-A-1 has been included in this book.

For best results, do the tutorials in the order in which they appear in the book. The tutorials are in the same order in which they appear on the menus of the electronic version.

If you have multiple tutorial books and you are new user of OPNET, it is a good idea to go through the tutorial books in the following order:

- 1) General Tutorials
- 2) <Product-specific> Tutorials (not available for all products)
- 3) ACE Tutorials and Examples

1 Introduction to IT Guru

Introduction

Overview

Welcome to IT Guru!

This tutorial teaches you the basics of using IT Guru. If you are new to IT Guru, this short introduction will help get you started.

If you are performing this tutorial for the first time, read the following pages for important information about using this tutorial and IT Guru.

About the Models

This tutorial was written for use with release 11.0.A of the IT Guru software and the May 2005 models. If you are using a different release, you might get different results or be unable to complete some lessons.

To complete the tutorials, you must install the IT Guru standard models, which include the required tutorial models. These models are normally installed when you install the Standard Model Library.

IT Guru standard models cover common protocols and vendor devices. The standard models are in the subdirectories under the OPNET release directory (<reldir>):

<reldir>\models\std*<protocol_name>*

<reldir> describes the directory that contains the current IT Guru software.

You can find your **<reldir>** by performing the following steps:

- 1 Select **Help > About This Application** in the main menu.
- 2 In the About OPNET IT Guru dialog box, click on the **Environment** tab, and then expand the **System Information** section.
- 3 Under System Information, find the **OPNET release directory**.

For example, the **<reldir>** for a default installation of this release of IT Guru on Windows is

C:\Program Files\OPNET\11.0.A

The tutorials use the Windows convention of the backslash character (\) as the separator in directory pathnames. If you are using Solaris, replace the backslash with a forward slash (/).

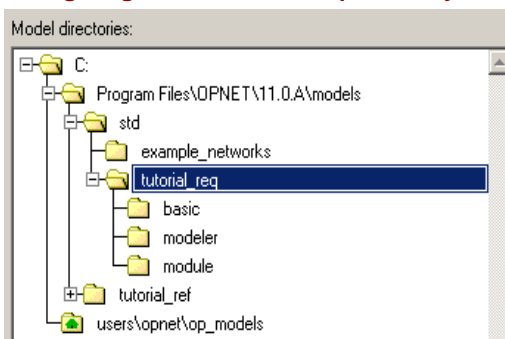
Opening Tutorial Files

Several tutorials will direct you to open an OPNET model file and save it with a unique prefix, such as your initials (<your_initials>). This way, several users can create and complete their own working copy of the same tutorial.

When you perform a tutorial, you are asked to open special tutorial model files. These model files are required to complete the tutorial; they are located in the **<reldir>\models\std\tutorial_req** directory or its subdirectories.

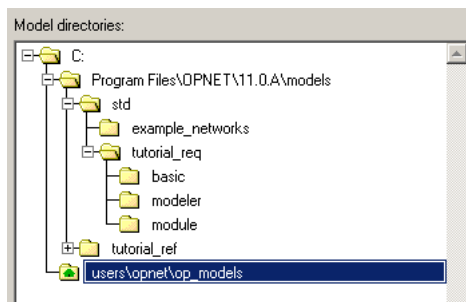
When you are asked to open model files that are NOT prefixed with your initials, make sure you navigate the directory structure in OPNET's open file browser (in the left pane) to the **tutorial_req** directory.

Navigating to the tutorial_req Directory



When you save a new model file, or use the **Save As...** command to save an existing model file with your initials, remember to navigate to the OPNET working directory (normally **<home>\op_models**) to save your files.

Navigating to the op_models Directory



About the Lessons

In this tutorial, you learn how to use the IT Guru features to build and analyze network models.

You can download the latest version of this tutorial from the OPNET Web site. Visit www.opnet.com/support/home.html and select the Product Updates link.

Each lesson presents a modeling problem for you to solve by building a network model, gathering statistical information about the network, and analyzing those results.

Each lesson helps you become familiar with IT Guru and demonstrates the range of problems IT Guru can solve.

We suggest that you perform the lessons and topics in sequence.

Most lessons have Key Concept paragraphs, like this one, that contain new information about IT Guru or describe important aspects of modeling theory.

Position this tutorial window next to IT Guru on your screen so that you can view both windows at the same time.

By default, the tutorial window always stays below the OPNET window. If you are using a Windows computer, you can change this behavior by setting the **itguru.tutorial_top** preference to **TRUE** (for information about changing preferences, see the Productivity Features lesson, Preferences topic).

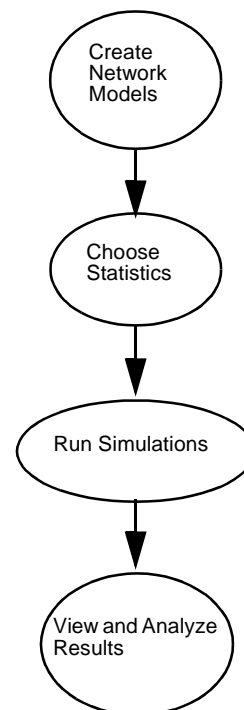
About IT Guru

Before you start, you should understand the IT Guru workflow, the workspace, and the editors. Become familiar with these essential features, and then begin the tutorial lessons.

The workflow for IT Guru (that is, the steps you use to build a network model and run simulations) centers around the Project Editor.

In this editor, you can create a network model, choose statistics to collect from each network object or from the whole network, execute a simulation, and view results.

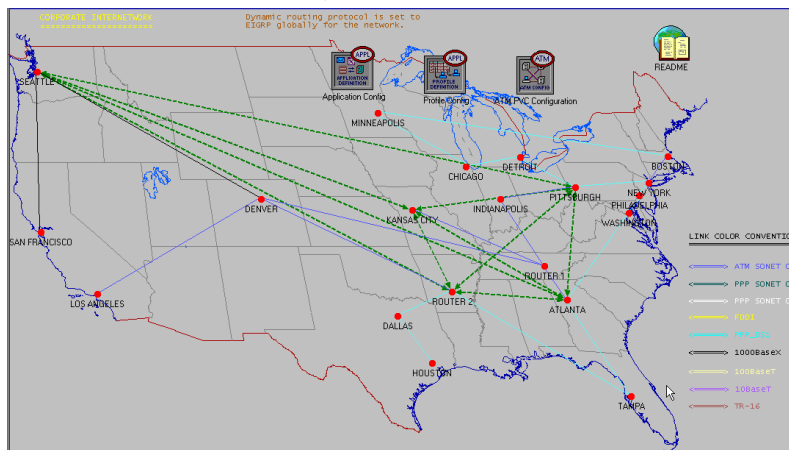
Your first look at IT Guru in Lesson 1 shows you how to use the Project Editor to build a small internetwork.



The Project Editor

The Project Editor is the main staging area for creating a network simulation. From this editor, you can build a network model using models from the standard library, choose statistics about the network, run a simulation, and view the results.

A Network Model in the Project Editor

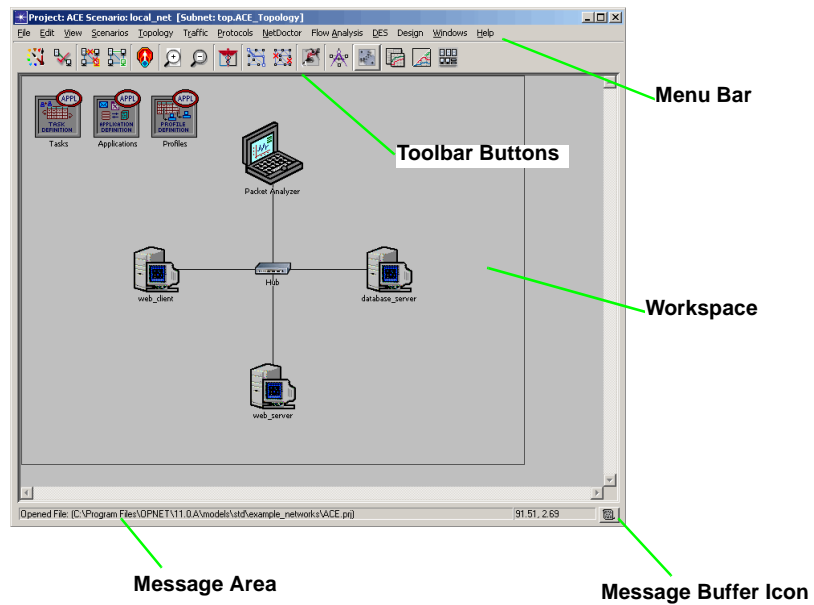


The Project Editor Window

There are several areas in the Project Editor window that are important for building and executing a model. These are shown on the following pages.

When you open an existing project, your screen should look similar to the following figure:

Project Editor Window



The Menu Bar

The menu bar is at the top of the editor window. It organizes all the non-context-sensitive editor operations into a set of topical menus.

The exact set of menus and menu operations available change according to the product modules that are enabled.

Context-sensitive editor operations are available by right-clicking on an object or on the background of the workspace.

Toolbar Buttons

Several of the more commonly used menu bar selections can also be activated through toolbar buttons. The following figure shows toolbar buttons that can appear in the Project Editor. You will use many of these toolbar buttons in the tutorial. (Note: Depending on your specific product and modules, you might not see all of the buttons shown here.)

Toolbar Buttons in the Project Editor



1 Open Object Palette	9 Configure/Run Flow Analysis
2 Verify Links	10 Configure/Run Failure Analysis
3 Fail Selected Objects	11 Configure/Run Discrete Event Simulation (DES)
4 Recover Selected Objects	12 Configure/Run Design Action
5 Go to Parent Subnet	13 Configure/Run Automation Tasks
6 Zoom to Rectangle	14 View Results
7 Zoom to Previous	15 Hide/Show Graph Panels
8 Configure/Run NetDoctor	16 Generate Network Inventory Summary

The Workspace

The central, scrollable region of the editor window is the workspace. This is where the network model appears; you can select and drag network objects in the workspace, and choose context-sensitive menu operations by right-clicking on the background.

The Message Area

The message area is located at the bottom of the editor window. It provides information about the status of the tool.

Message Area

No reports have been generated for the project (Frame_Relay) scenario (attr_based_pvc).

You can left-click on the icon next to the message area to open the message buffer window.



The message buffer window shows a list of messages that have appeared in the message area. You can open the message buffer window if part of the message line is truncated in the message area or if you think a later message has replaced an important alert or notification.

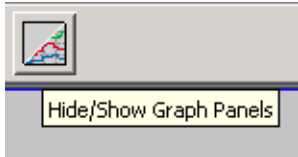
Tooltips

If you rest your cursor over a tool button or a network object in the workspace, a help balloon appears. The tooltip describes one of the following:

- the action that occurs if the button is pressed
- information about the network object



Tooltip



Documentation

Occasionally, you might have questions about certain aspects of the tutorial, the tool, or the models. You can refer to these sources:

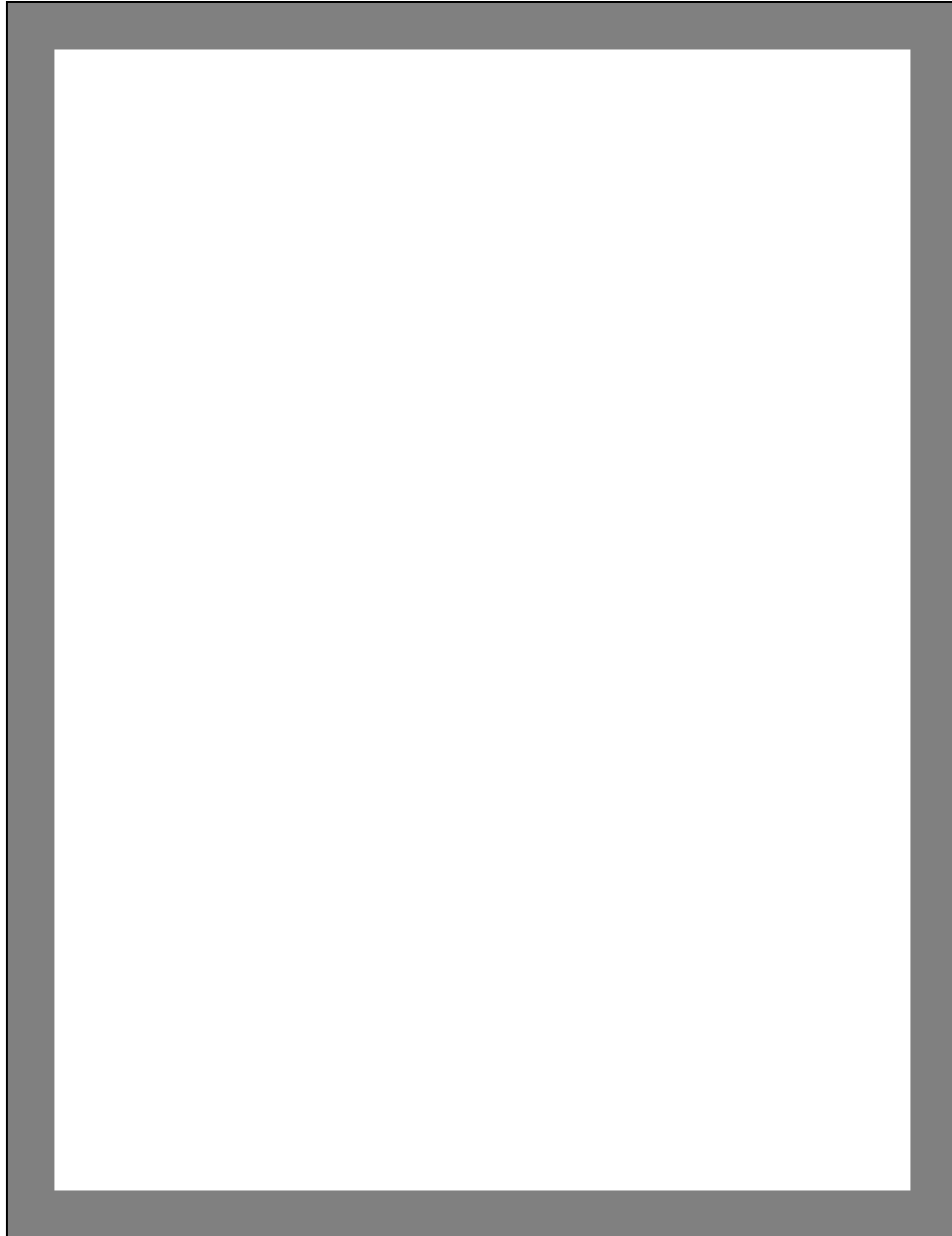
- **Built-in documentation** is available throughout the tool. For example, you can get model descriptions, attribute definitions, and statistic descriptions by selecting an item and clicking on the **Details** button. Some dialog boxes also have Help buttons that bring up additional descriptive information.
- **OPNET product documentation.** Select **Help > Product Documentation**.
- **FAQ** (Frequently Asked Questions). Point your browser to **www.opnet.com/support**.

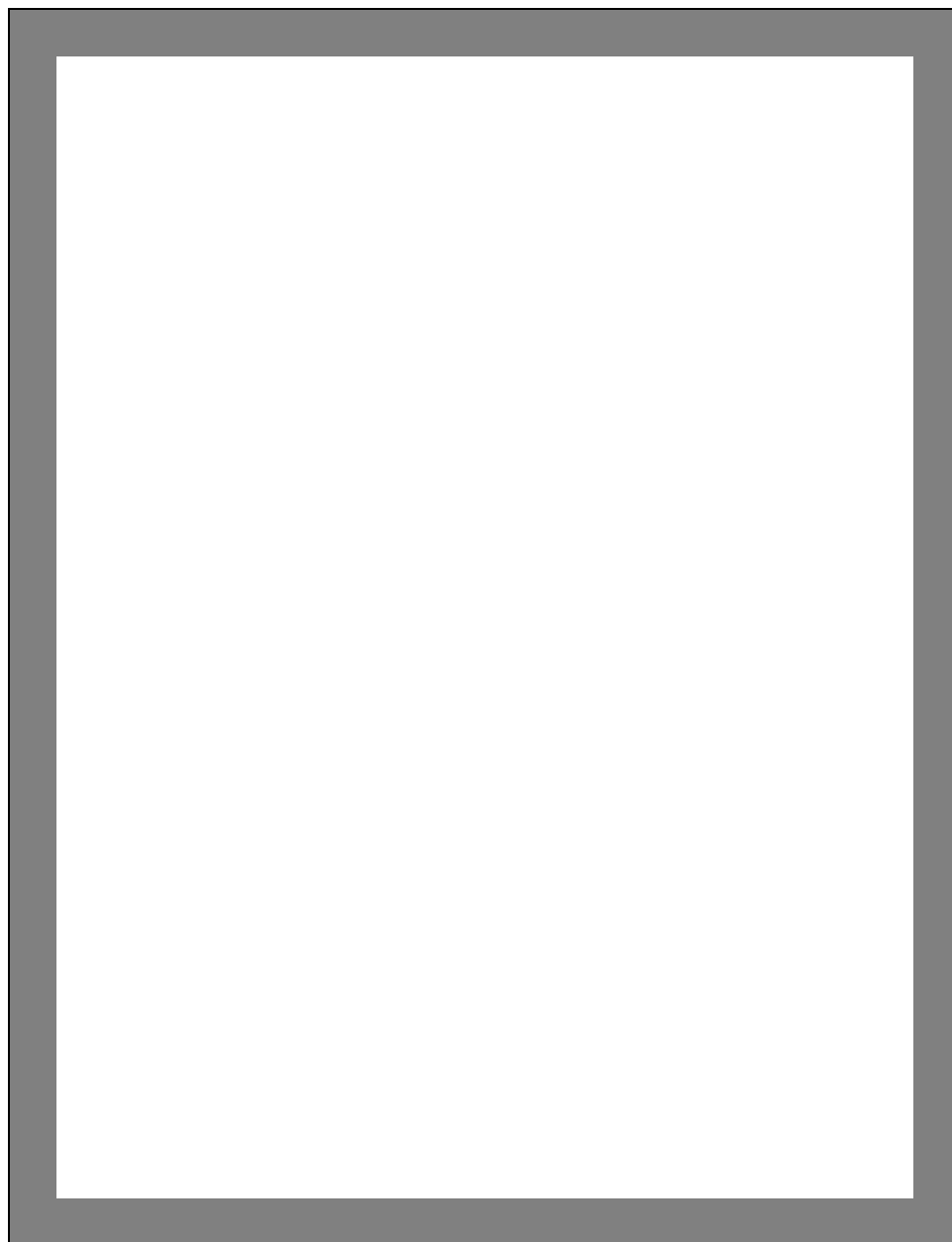
You need your user name and password to access the Web FAQ. OPNET Technical Support normally sends your user name and password by email after you register your OPNET software for purchase or evaluation.

- **Recent updates** to the tutorial lessons and models can be downloaded from the Web site: **www.opnet.com/support**.

- **Model User Guides** provide the latest information on the protocol models and how to use them. In the Project Editor window, choose **Protocols > <protocol name> > Model User Guide**.

Now you are ready to begin the tutorial. Return to the main tutorial menu and choose **Small Internetworks** from the list of available lessons. Have fun learning IT Guru!





2 Small Internetworks

Small Internetworks

Introduction

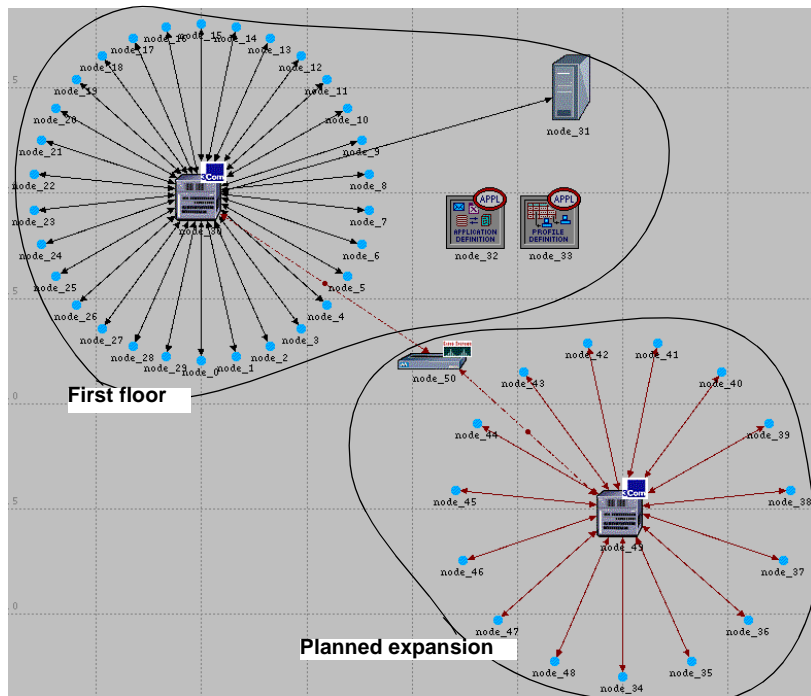
In this lesson, you will see how IT Guru can do organizational scaling to solve a typical “what if” problem. You will learn how to use IT Guru features to build and analyze network models. In this lesson, you will

- Build a network quickly
- Collect statistics about network performance
- Analyze these statistics

In this lesson, you use the Project Editor to build a topology of a small internetwork, choose statistics to collect, run a simulation, and analyze the results.

In this lesson, you plan for the expansion of a small company’s intranet. Currently, the company has a star topology network on the first floor of its office building and plans to add an additional star topology network on another floor. You will build and test this “what-if” scenario to ensure that the load added by the second network will not cause the network to fail.

The Final Network



Getting Started

When creating a new network model, you must first create a new **project** and **scenario**. A project is a group of related scenarios that each explore a different aspect of the network. Projects can contain multiple scenarios.

After you create a new project, you use the Startup Wizard to set up a new scenario. The options in the Wizard let you

- Define the initial topology of the network
- Define the scale and size of the network
- Select a background map for the network
- Associate an object palette with the scenario

The **Startup Wizard** automatically appears each time you create a new project. The Startup Wizard lets you define certain aspects of the network environment.

To use the Startup Wizard to set up a new scenario, do the following:

- 1 If IT Guru is not already running, start it.
- 2 Select **File > New...**
- 3 Select **Project** from the pull-down menu and click **OK**.
- 4 Name the project and scenario, as follows:
 - 4.1 Name the project **<initials>_Sm_Int**

Include your initials in the project name to distinguish it from other versions of this project.
 - 4.2 Name the scenario **first_floor**.
 - 4.3 Click **OK**.
 - The Startup Wizard opens.
- 5 Enter the values shown in the following table in the dialog boxes of the Startup Wizard.

Values to Enter in the Startup Wizard

Dialog Box Name	Value
1. Initial Topology	Select the default value: Create empty scenario.
2. Choose Network Scale	Select Office . Select the Use metric units checkbox.
3. Specify Size	Select the default size: 100 m x 100 m
4. Select Technologies	Include the Sm_Int_Model_List model family.
5. Review	Check values, then click Finish .

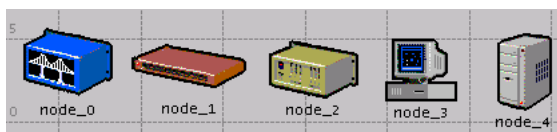
- ➔ A workspace of the size you specified is created. The object palette you specified opens in a separate window.

Creating the Network

Network models are created in the Project Editor using **nodes** and **links** from the **object palette**.

Node—A representation of a real-world network object that can transmit and receive information.

Nodes



Link—A communication medium that connects nodes to one another. Links can represent electrical or fiber optic cables.

A Link



These objects are found in the **object palette**, a dialog box that contains graphical representations of node and link models.

If it is still open, close the object palette.

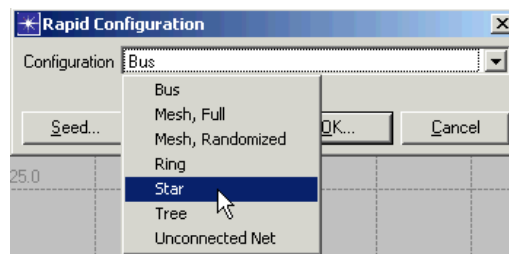
You can use any of three methods to create a network topology, or a combination of all three. One method is to import the topology (discussed in a later lesson). Another is to place individual nodes from the object palette into the workspace. The third method is to use **Rapid Configuration**.

Rapid Configuration creates a network in one action after you select a network configuration, the types of nodes within the network, and the types of links that connect the nodes.

To create the first-floor network using Rapid Configuration:

- 1 Select **Topology > Rapid Configuration**.
- 2 Select **Star** from the pull-down menu of available configurations, then click **OK...**

Available Configurations Pull-Down Menu



Specify the node models and link models in the network. Models follow this naming scheme:

<protocol 1>_..._<protocol n>_<function>_<mod>

where:

- **<protocol>** specifies the specific protocol(s) supported by the model
- **<function>** is an abbreviation of the general function of the model
- **<mod>** indicates the level of derivation of the model

For example:

ethernet2_bridge_int

specifies the intermediate (**int**) derivation of a 2-port Ethernet (**ethernet2**) bridge (**bridge**).

Vendor models have an additional prefix that specifies the vendor and the vendor product number for that particular network object.

For example, the 3Com switch used in this lesson is named:

3C_SSII_1100_3300_4s_ae52_e48_ge3

This node is a stack of two 3Com SuperStack II 1100 and two Superstack II 3300 chassis (**3C_SSII_1100_3300**) with four slots (**4s**), 52 auto-sensing Ethernet ports (**ae52**), 48 Ethernet ports (**e48**), and 3 Gigabit Ethernet ports (**ge3**).

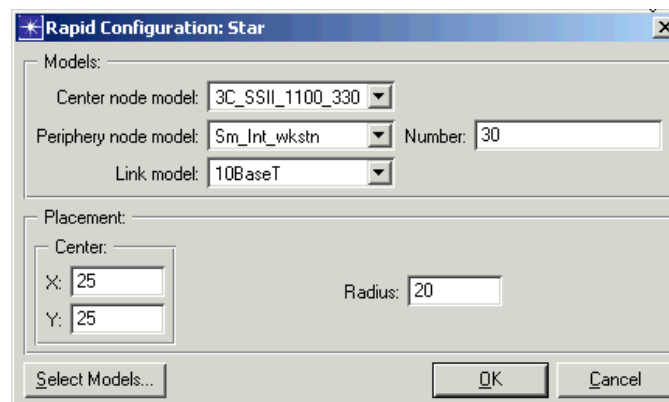
To specify the nodes and links to use to build the network:

- 1 Set the **Center Node Model** to **3C_SSII_1100_3300_4s_ae52_e48_ge3**. This is a 3Com switch.
- 2 Set the **Periphery Node Model** to **Sm_Int_wkstn**, and change the **Number** of periphery nodes to **30**. This sets 30 Ethernet workstations as the peripheral nodes.
- 3 Set the **Link Model** to **10BaseT**.

Specify where the new network will be placed:

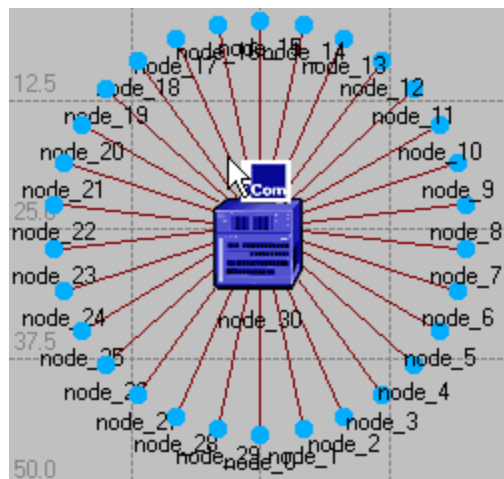
- 1 Set the **X center** and **Y center** to **25**.
- 2 Set the **Radius** to **20**.

Rapid Configuration Dialog Box



- 3 Click **OK**.
 - ➔ The network is drawn in the Project Editor.

The First Floor Network



Now that the general network topology has been built, you need to add a server. You will use the second method of creating network objects: dragging them from the object palette into the workspace.

- 1 If it is not already open, open the object palette by clicking on the **Object Palette** toolbar button.
- 2 Find the **Sm_Int_server** object in the palette and drag it into the workspace.



You will not find this exact server model on other object palettes because we created it with the correct configuration for this tutorial.

By default, you can create additional instances of the same object by left-clicking after the initial “drag-and-drop” from the palette.

- 3** Because you do not need additional copies of this model, right-click to turn off node creation.

You also need to connect the server to the star network.

- 1 Find the **10BaseT** link object in the palette and click on it.
- 2 Click on the server object, then click on the switch object in the center of the star.
 - A link is drawn, connecting the two objects.
- 3 Right-click to turn off link creation.

Finally, you need to add configuration objects to specify the application traffic that will exist on the network. Configuring the application definition and profile definition objects can be complicated, so you do not have to do these tasks right now. For this tutorial, we included, on the object palette:

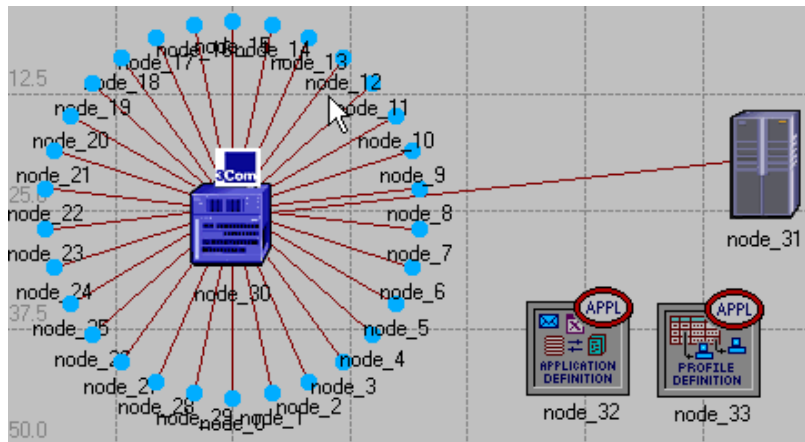
- an application definition object with the default configurations of the standard applications, and
- a profile definition object with a profile that models light database access

You need only drag the objects into your network. Doing so means that the traffic caused by workstations accessing a database at a low rate will be modeled.

- 1 Find the **Sm_Application_Config** object in the palette and drag it into the workspace
- 2 Right-click to turn off object creation.
- 3 Find the **Sm_Profile_Config** object in the palette, drag it into the workspace, and right-click.
- 4 Close the object palette.

The network is now built and should look similar to the following figure.

The Finished First Floor Network



You are now ready to begin collecting statistics.

Collecting Statistics

You can collect statistics from individual nodes in your network (**object statistics**) or from the entire network (**global statistics**).

Now that you have created the network, you should decide which statistics you need to collect to answer the questions presented earlier in this lesson:

- Will the server be able to handle the additional load of the second network?
- Will the total delay across the network be acceptable once the second network is installed?

To answer these questions, you need a snapshot of current performance for comparison. To get this baseline, you will collect one object statistic, **Server Load**, and one global statistic, **Ethernet Delay**.

Server load is a key statistic that reflects the performance of the entire network. To collect statistics related to the server's load, do the following steps:

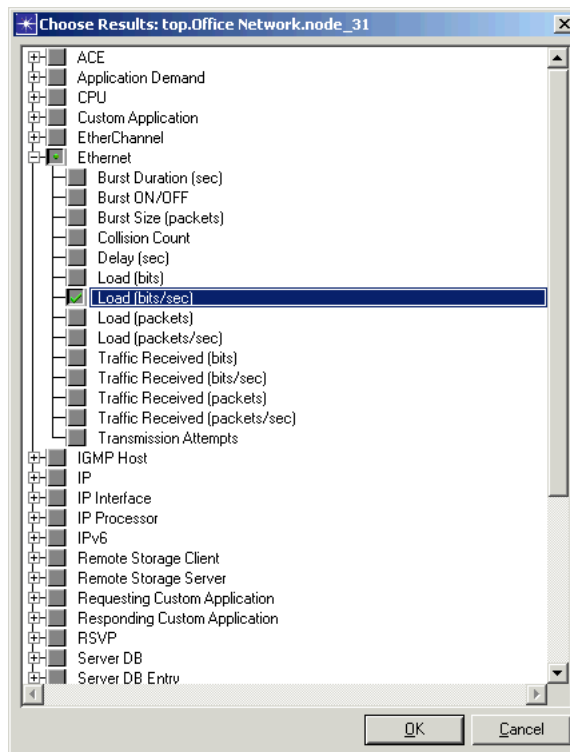
- 1 Right-click on the server node (**node_31**) and select **Choose Individual DES Statistics** from the server's Object pop-up menu.

➔ The Choose Results dialog box for node_31 appears.

The Choose Results dialog box hierarchically organizes the statistics you may collect. To collect the Ethernet load on the server:

- 2 Expand the treeview for **Ethernet** in the **Choose Results** dialog box to see the Ethernet statistic hierarchy.

Choose Results Dialog Box

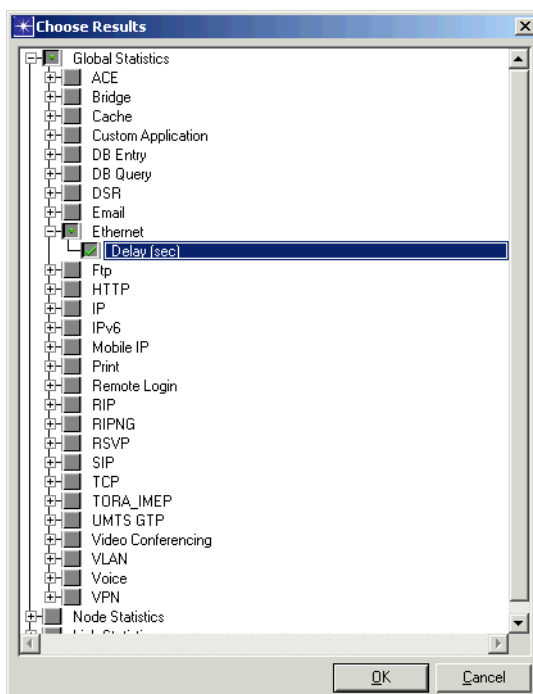


- 3 Click the checkbox next to **Load (bits/sec)** to enable collection for that statistic.
- 4 Click **OK** to close the dialog box.

Global statistics can be used to gather information about the network as a whole. For example, you can find out the delay for the entire network by collecting the global **Delay** statistic:

- 1 Right-click in the workspace (but not on an object) and select **Choose Individual DES Statistics** from the Workspace pop-up menu.

Global Statistic Chosen



- 2 Expand the Global Statistics hierarchy.
- 3 Expand the Ethernet hierarchy.
- 4 Click the checkbox next to **Delay (sec)** to enable data collection.
- 5 Click **OK** to close the Choose Results dialog box.

It is good to get into the habit of saving your project every so often. To save the project:

- 1 Choose **File > Save**, then click **Save** (the project already has a name, so you don't need to rename it).

Now that you have specified the statistics to collect and saved the project, you are almost ready to run your simulation.

First, though, verify that your **repositories** preference is set appropriately.

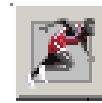
- 1 Choose **Edit > Preferences**.
- 2 Type **repositories** in the **Find** field and click on the **Find** button.

- 3 If the value for **repositories** is not **stdmod**, click on the field.
- 4 When the **repositories** dialog box appears, click the Insert button then enter **stdmod** in the dialog box.
- 5 Click **OK** to close the **repositories** and **Preferences** dialog boxes.

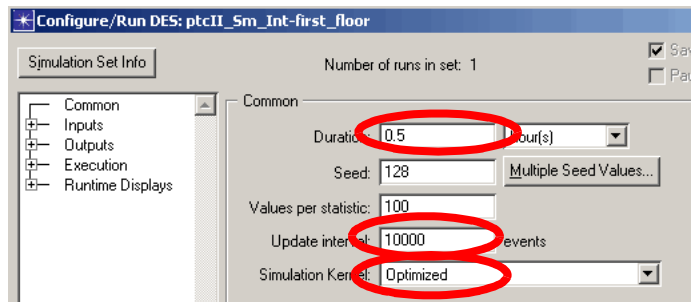
To run a simulation:

- 1 Select **DES > Configure/Run Discrete Event Simulation....**

You can also open the Configure Discrete Event Simulation dialog box by clicking on the **Configure/Run Discrete Event Simulation (DES)** toolbar button.



Configure Discrete Event Simulation Dialog Box



- 2 Type **0.5** in the **Duration:** field to simulate one-half hour of network activity.

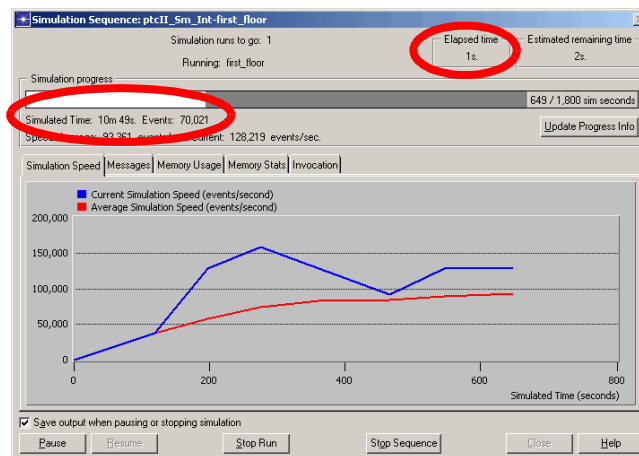
In this case, the simulation calculates and displays events/second data at 10,000-event intervals. The default setting for this is 500,000 for larger network simulations.

- 3 Click the **Run** button to begin the simulation.

While the simulation runs, a dialog box appears showing the simulation's progress.

Simulation Sequence Dialog Box

Elapsed Time: Number of seconds the simulation has run



Simulated Time: Minutes of network time

The dialog box above shows that, in 1 second of elapsed (actual) time, IT Guru has simulated 10 minutes and 49 seconds of network time. The entire simulation should take less than one minute to complete—the elapsed time varies according to the speed of your computer.

- 4 When the simulation finishes, the contents of the Messages tab appears. Click the **Close** button in the Simulation Sequence dialog box.
- 5 If your simulation does not complete, if no results were collected, or if the results vary significantly from those shown, you will have to troubleshoot your simulation. See "*Troubleshooting IT Guru Tutorials*".

Viewing Results

You can view results graphically in the Project Editor by selecting **View Results** from the Workspace pop-up menu.

After your simulation has executed, you will want to see the information collected for each statistic. There are several ways to view results; in this lesson you will use the View Results option in the Workspace pop-up menu.

You will learn different ways to view results in later lessons.

To view the server Ethernet load for the simulation:

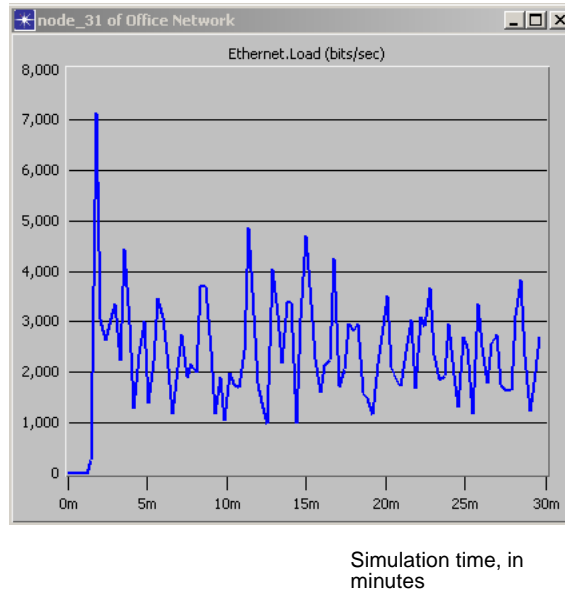
- 1 Right-click on the server node (**node_31**) choose **View Results** from the server's Object pop-up menu.
➔ The node's View Results dialog box opens.
- 2 Expand the **Office network.node_31 > Ethernet** hierarchy.
- 3 Click on the checkbox next to **Load (bits/sec)** to indicate that you want to view that result.

- 4 Click the **Show** button in the View Results dialog box.
 - ➔ The graph of the server load appears in the Project Editor, as shown in the following figure.

The graph of the server load should resemble the following graph. Your results may differ slightly due to differences in node placement and link length, but the general trends should be consistent.

Server Load Graph

bits/second. The unit of measure on this axis is shown in the statistic in the Choose Results dialog box



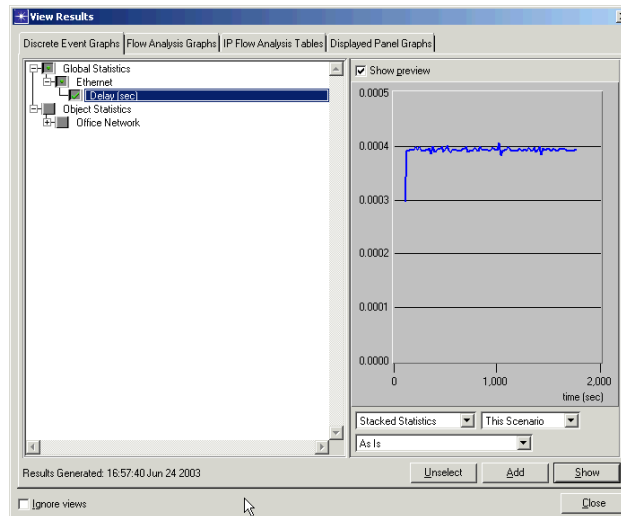
Note that at its peak, the load on the server is about 7,000 bits/second. You will need this baseline for comparison after you add the second network.

When you finish viewing the server load graph, close this dialog box and the View Results dialog box. (If the system prompts you, choose to delete the graph panel.)

You also should look at the Global Ethernet Delay on the network. To view this statistic:

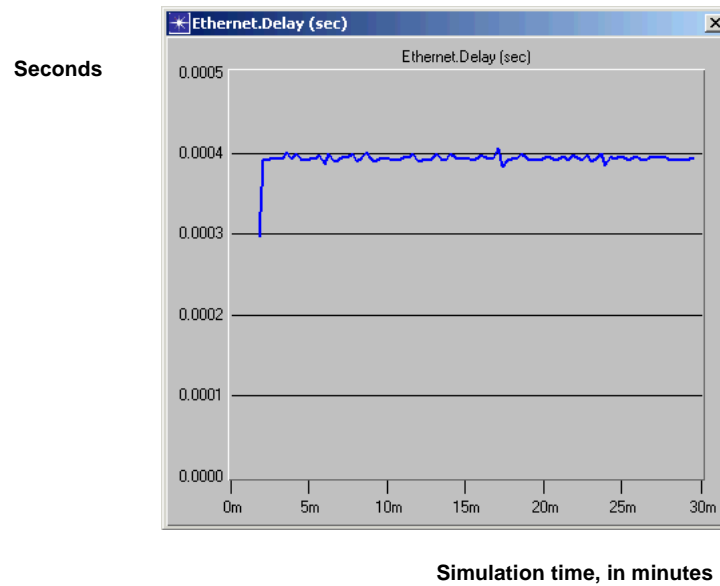
- 1 Right-click in the workspace, then select **View Results** from the pop-up menu.
- 2 Check the box next to **Global Statistics > Ethernet > Delay (sec)**, then click the **Show** button to view the Ethernet delay for the whole network.

Viewing Ethernet Delay for the Whole Network



➔ The Ethernet delay graph appears in the Project Editor. The graph should resemble the following figure.

Ethernet Delay Graph



Note that after the network reaches steady state the maximum delay is around 0.4 milliseconds.

When you finish viewing the graph, close the graph and the View Results dialog box.

Expanding the Network

You have created a baseline network and gathered statistics about it. Now you are ready to expand the network and verify that it still operates sufficiently well with the additional load.

When performing a “what-if” comparison, it is convenient to store the baseline network as one scenario and create the experimental network as a different scenario. You will duplicate the existing scenario and make changes to it instead of building the new topology from the beginning.

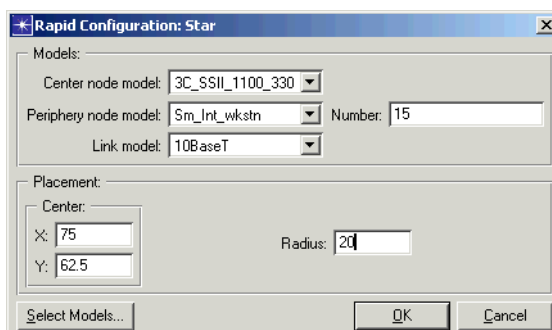
To duplicate a scenario:

- 1 Choose **Scenarios > Duplicate Scenario...**
- 2 Enter **expansion** as the name for the new scenario.
- 3 Click **OK**.
 - ➔ The scenario, with all the nodes, links, statistics, and the simulation configuration, is duplicated and named **expansion**.

The second-floor segment will resemble the first-floor segment, but will not have a server of its own. To build the new segment:


- 1 Select **Topology > Rapid Configuration**.
- 2 Choose **Star** for the topology and click **OK...**
- 3 Complete the Rapid Configuration dialog box with these values:
 - Center Node Model:
3C_SSII_1100_3300_4s_ae52_e48_ge3
 - Periphery Node Model: **Sm_Int_wkstn**
 - Number: **15**
 - Link model: **10BaseT**
 - X: **75**, Y: **62.5**, Radius: **20**

Rapid Configuration Dialog Box



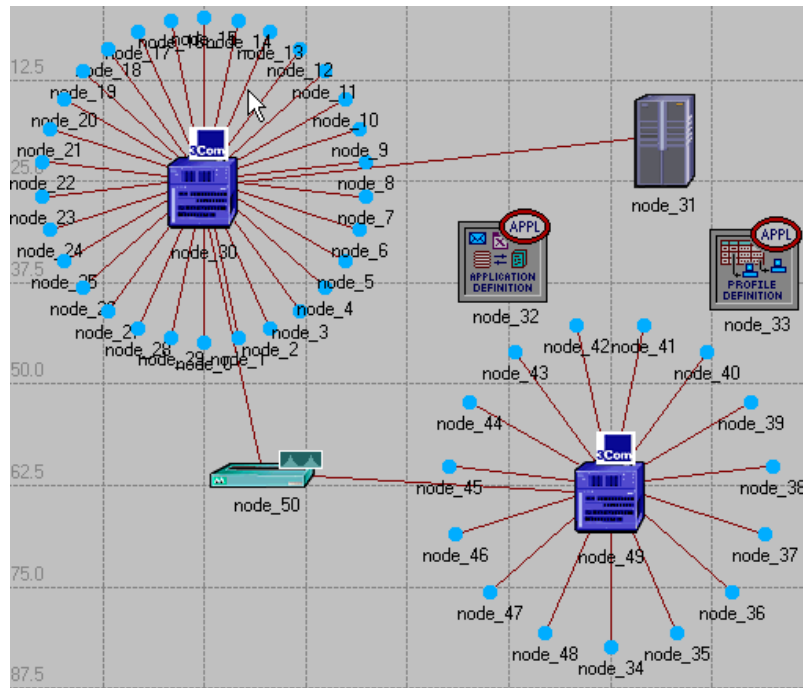
- 4 Click **OK** to create the network.

Join the two networks:

- 1 If it is not already open, click the toolbar button to open the object palette. 
- 2 Drag the **Cisco 2514** router icon into the workspace between the two networks. Right-click to turn off node creation.
- 3 Click on the **10BaseT** link icon in the object palette.
- 4 Create **10BaseT** links between the Cisco router (**node_50**) and the 3Com switches at the center of each star.
- 5 Right-click to turn off link creation.
- 6 Close the object palette.
- 7 Select **File > Save**.

The final network should look like this:

The Final Network

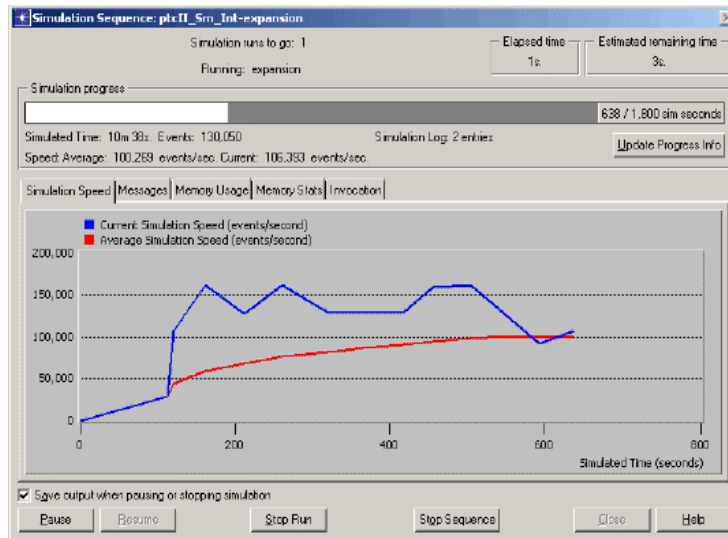


To run the expansion scenario:

- 1 Select **DES > Configure/Run Discrete Event Simulation....**
- 2 Verify that the **Duration** is set to **0.5** hours and the **Update interval** is set to **10000**.

3 Click the **Run** button to begin the simulation.

Simulation Sequence Dialog Box, Simulation Speed Tab Selected



➔ As before, a window displays simulation start-up messages first, and then an animated graph shows both the current and average speed in events per second during the simulation. When the simulation is completed, you can view the event/second graph results from the Simulation Speed tab.

- 4 When the simulation is done, close the Simulation Sequence dialog box. If you had problems, see "*Troubleshooting IT Guru Tutorials*".

Comparing Results

To answer the questions posed about the addition of a second network to the existing LAN, you need to compare the results from both of the simulations you ran.

You will use the **Compare Results** menu item in the Object and Workspace pop-up menus to combine statistics from different scenarios in the same graph.

To look at the server load from both scenarios at once:

- 1 Right-click on the server node (**node_31**) to bring up its Object pop-up menu.
- 2 Choose **Compare Results** (you can do this from either scenario in the project).
 - ➔ A Compare Results dialog box appears containing a hierarchical representation of statistics collected for the server node.

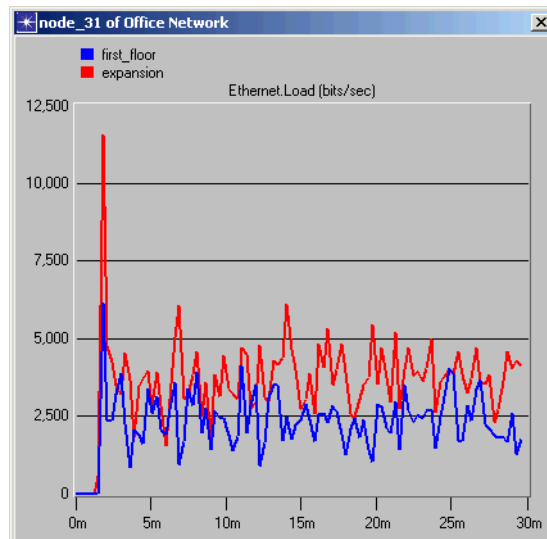
If your results differ radically from those shown in the following figures, you will have to troubleshoot your simulation. See "*Troubleshooting IT Guru Tutorials*".

When comparing results, choosing a statistic in one scenario produces a graph showing the value of that statistic in all scenarios. To view the results:

- 1 Select the **Office Network.node_31 > Ethernet > Load (bits/sec)** statistic and click the **Show** button. Your results should resemble those in the following figure (but may not be identical):

Server Load Compared

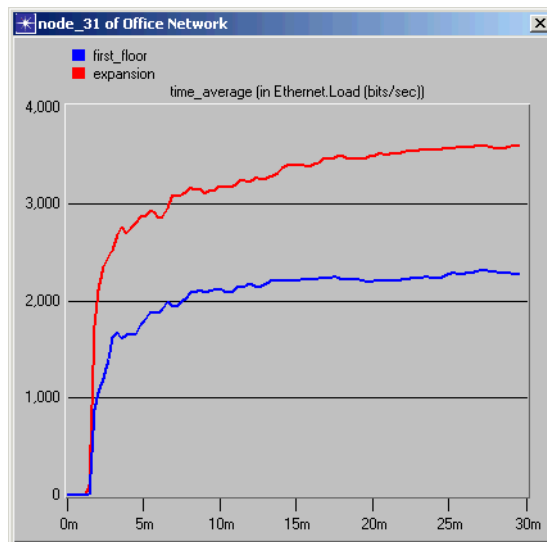
bits/second



Simulation time, in minutes

The following graph is the time average of the Ethernet load between the baseline (first_floor) scenario and the expansion scenario. You will learn how to create a graph of the time average in the next lesson.

Time-averaged Server Load Compared



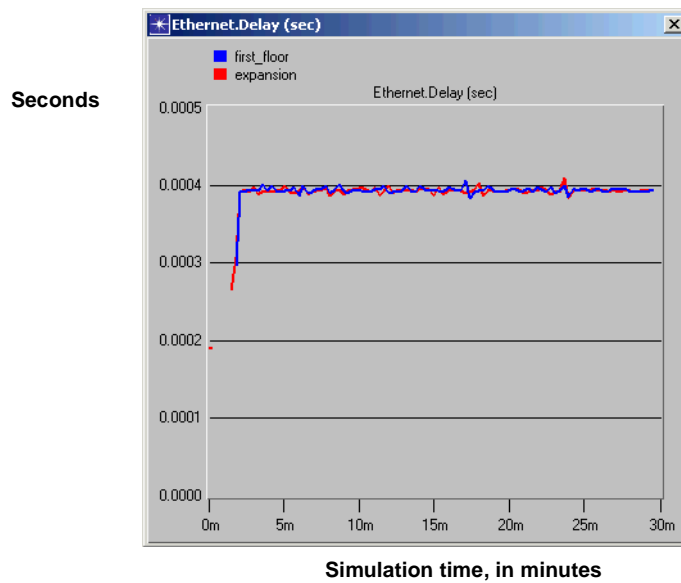
Note that while the average load for the expansion scenario is higher (as expected), the load as a whole appears to be leveling off (that is, not monotonically increasing), indicating a stable network.

The last step is to see how much the network's delay is affected by adding a second floor. To compare Ethernet delay for the two scenarios:

- 1 Close the graph and the Compare Results dialog box for the server.
- 2 Right-click in the workspace, then select **Compare Results** from the Workspace pop-up menu.
- 3 Select the **Global Statistics > Ethernet > Delay (sec)** statistic.
- 4 Click **Show** to display the graph.
 - ➔ The graph of the Global Ethernet Delay appears.

Your graph of Ethernet Delay should resemble the following:

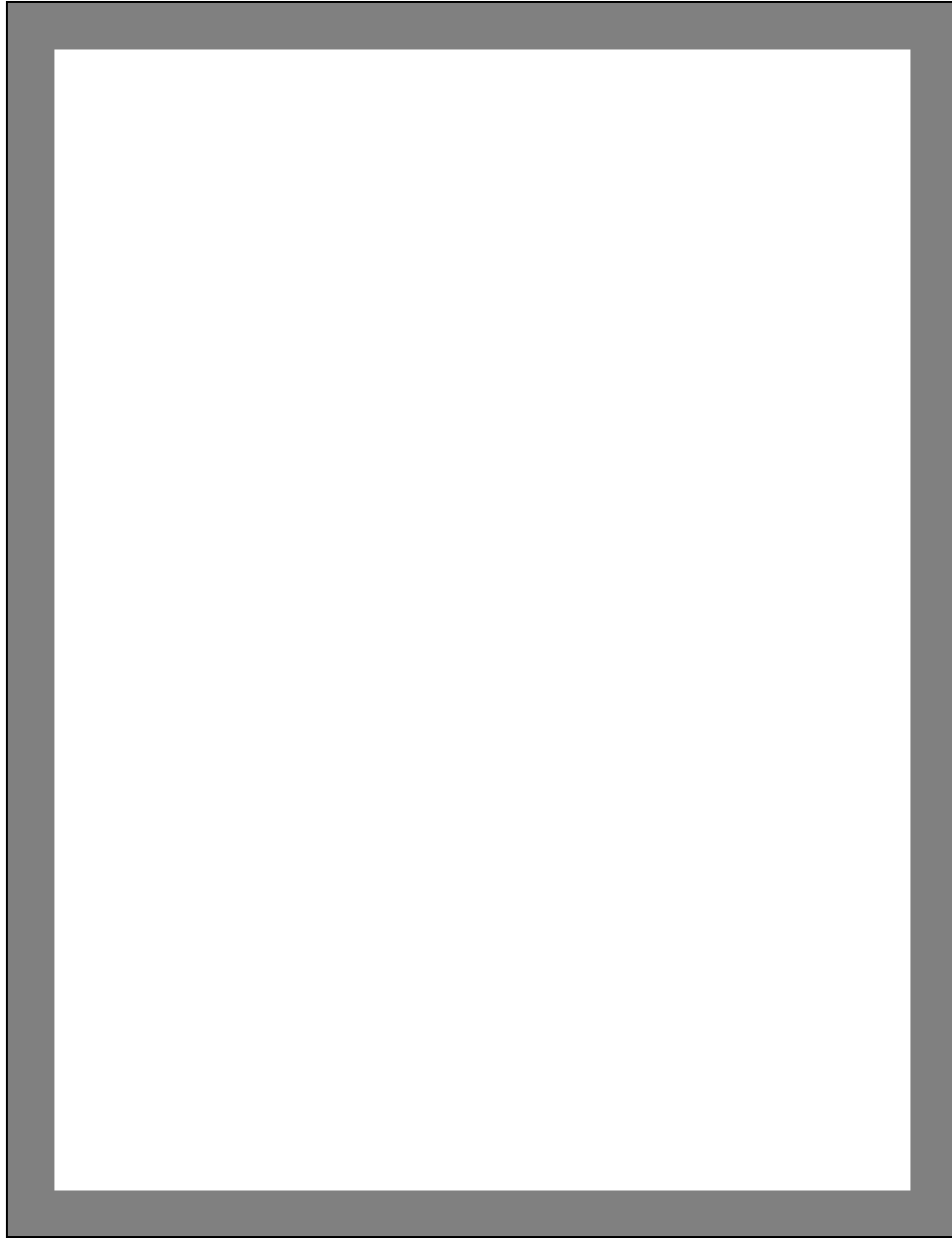
Ethernet Delay Compared

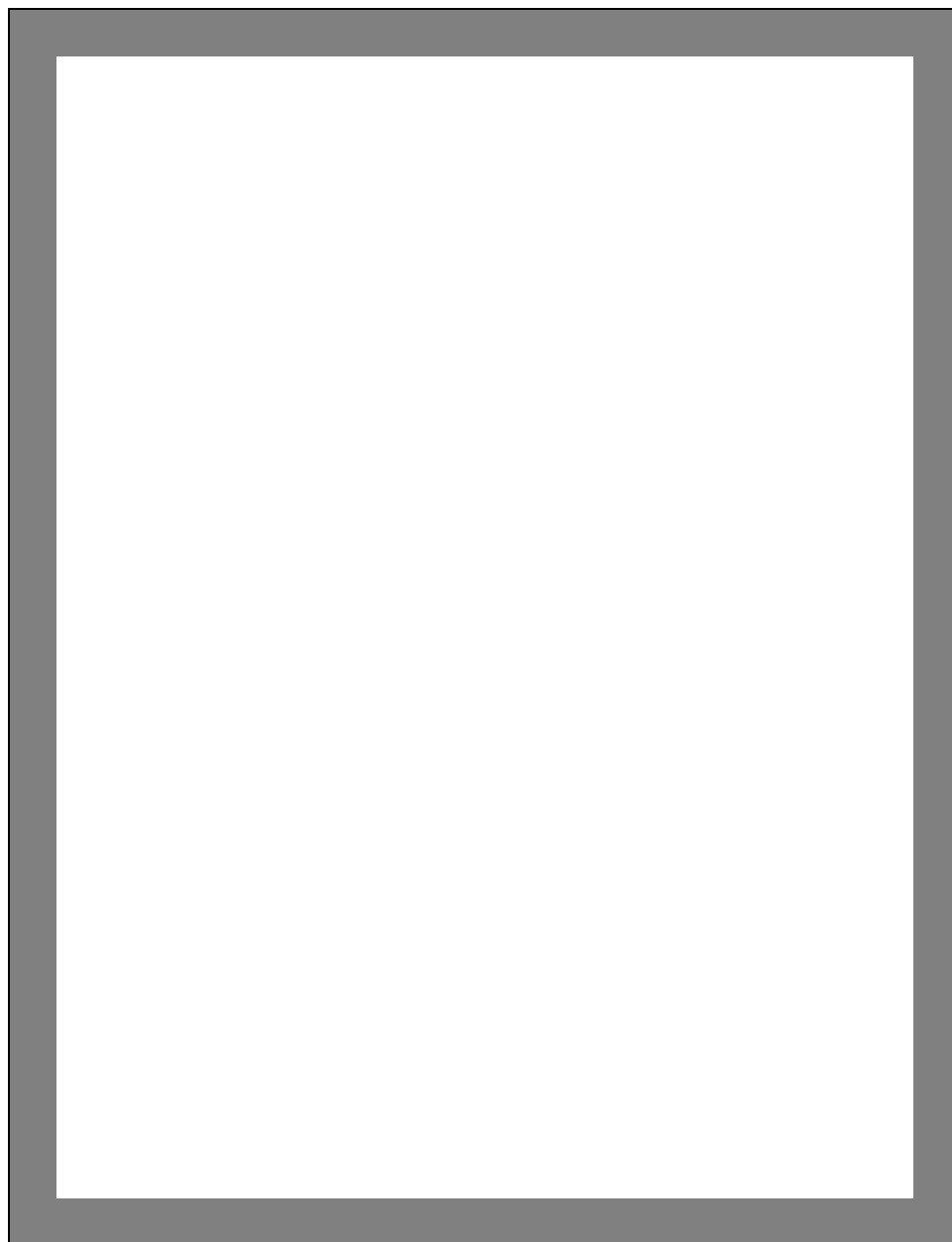


This graph shows that there is no significant change in Ethernet delay on the network. Although server load has increased, delay has not.

- 5 Close the open graph and the **View Results** dialog box.
- 6 Select **File > Close** and save changes before closing.

Now you are ready to go on to the LAN Networks tutorial. This lesson explores the use of LAN objects in building networks, as well as background traffic. Return to the main tutorial menu and choose **LAN Networks** from the list of available lessons.





3 LAN Modeling

LAN Modeling

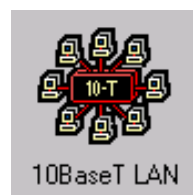
Overview

This lesson focuses on the use of LAN models and background link utilization. You will learn how to

- Configure the object palette with the models you need
- Set up application and profile configurations
- Model a LAN as a single node
- Specify background utilization that changes over time on a link
- Simulate multiple scenarios simultaneously
- Apply filters to graphs of results and analyze the results

This lesson focuses on two features, **LAN models**, which model entire LANs, and the link **Background Load** attribute, which is used to model existing link traffic.

- **LAN Models:** If you are modeling a medium or large internetwork, you may only be interested in specific aspects of the network's behavior (whether a key router will be overloaded, for example). In such cases, single nodes can model entire LANs.
- **Link Background Load:** Use the **Background Load** attribute to model existing traffic on a link instead of explicitly modeling each packet. You can also specify changes in this background traffic over the course of a simulation.



Background Load Attribute Dialog Box

The dialog box displays a table of background load attributes and a graph of the load profile. The table lists attributes such as Average Packet Size and Intensity with their respective values. The graph shows a load profile over time, with a peak around 3,000 seconds.

Attribute	Value
Average Packet Size [bytes] [Atlanta-Core ...]	Default
Average Packet Size [bytes] [SF-Access -> ...]	Default
Intensity [bps] [Atlanta-Core -> SF-Access]	4

seconds	bits/second
0.0	289,349.333333333
600	455,450.666666667
1,200	488,218.666666667
1,800	520,986.666666667
2,400	553,754.666666667
3,000	557,056
3,600	589,824
4,200	557,056
4,800	553,754.666666667
5,400	520,986.666666667
6,000	455,450.666666667

Setting Up the Scenario

In this lesson, your job is to model an east coast company's WAN. The company has offices in Atlanta, Philadelphia, New York, and Boston, which are connected to the central network in Washington, D.C. The offices use phone lines to connect to each other, and are therefore susceptible to delays caused by additional, unrelated traffic on the lines.

This company wants you to determine how this background traffic is affecting FTP traffic on their network. To do this, you will model FTP performance on the network, first without background traffic and then with background traffic. Because you are not interested in modeling the details of each office's LAN, you will use LAN models to model the individual LANs as single nodes.

The first step in setting up the WAN is to specify the overall context for the network with the Startup Wizard. Once that is done, you can proceed with building the network itself. This topic focuses on:

- Configuring an object palette
- Specifying a map background
- Zooming in on a background

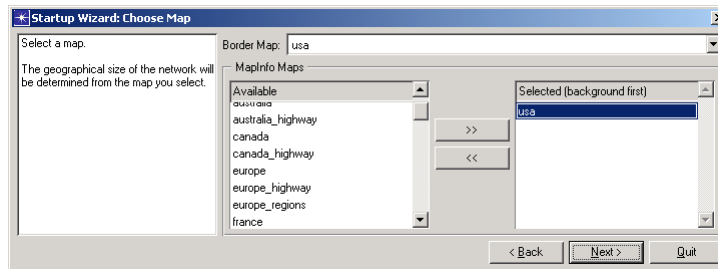
Begin by opening a new project in IT Guru and configuring the scenario context using the Startup Wizard:

- 1 If is not already running, start it, select **File > New... > Project**, then click **OK**.
- 2 Name the new project **<initials>_LAN_Mod** and the scenario **no_back_load**, then click **OK**.
- 3 Click **Next >** when the Startup Wizard opens and select “Create empty scenario” for the **Initial Topology**.

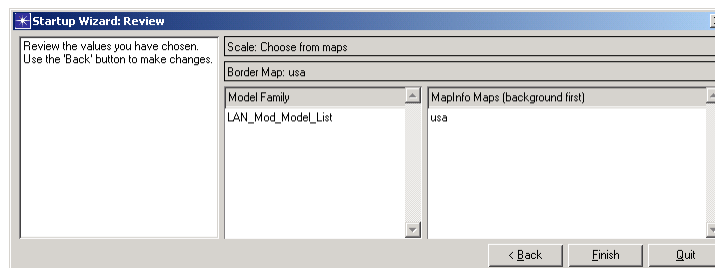
You can specify a map to use as a background for your network. To set a map background:

- 1 Click **Choose from maps** for **Network Scale** and click **Next >**.
- 2 Choose **usa** from the **Border Map** list.
- 3 Choose **usa** from the **MapInfo Maps** list and click **>>** to move it to the **Selected (background first)** pane.

Choose Map Wizard



- 4 Click **Next >**.
- 5 Select **LAN_Mod_Model_List** to be included in your network by clicking on the **Include?** cell and changing the value from **No** to **Yes**. Click **Next >**.
- 6 Review your settings and click **Finish** to close the Startup Wizard.



- ➔ The workspace now shows the specified map and object palette.

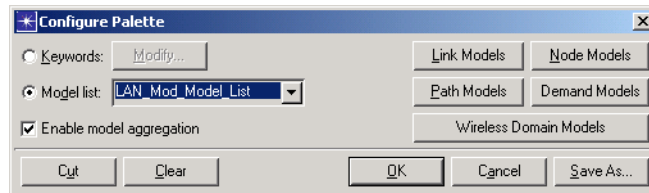
IT Guru's full set of node and link models would be overwhelming to work with all at once, so the object palette can be configured to show only a specific subset, or model list. You can use the standard model lists, adapt them for your own needs, or make your own list.

You can create your own custom object palettes for use in modeling projects.

For this lesson, we created the **LAN_Mod_Model_List**. Now you will adapt that model list by adding the LAN node model to it:

- 1 Click the **Configure Palette...** button in the object palette.
➔ The Configure Palette dialog box opens.

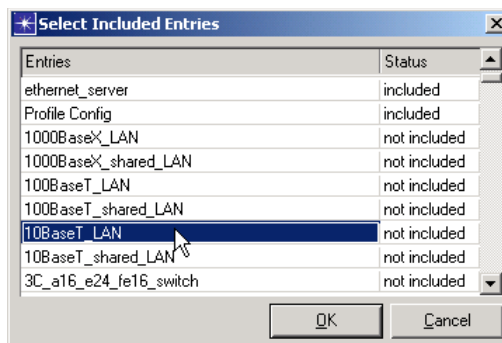
Configure Palette Dialog Box



The Configure Palette dialog box lets you change the object palette and then save it.

- 1 Click the **Node Models** button in the Configure Palette dialog box.
- 2 Find **10BaseT_LAN** in the Select Included Entries dialog box and change its status from **not included** to **included**.

10BaseT LAN Added to Object Palette

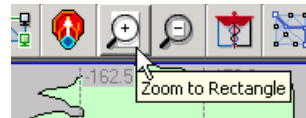


- 3 Click **OK**.
 - ➔ The **10BaseT_LAN** icon appears in the object palette.
- 4 Click **OK** to close the Configure Palette dialog box, then click **Save As** to save the model list as **<initials>_LAN_Mod-no_back_load**.

- 5 Click **OK** to close the dialog box.

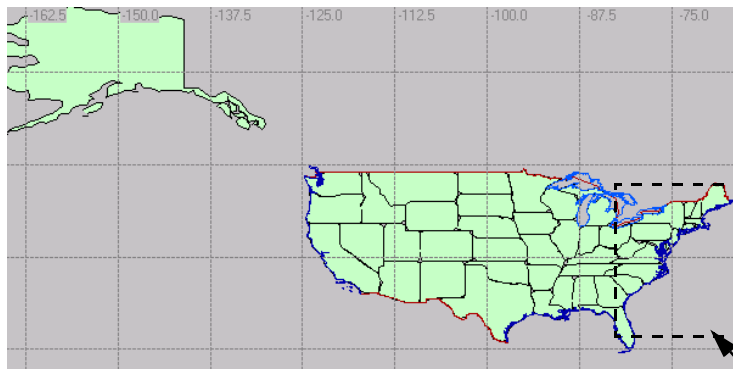
Because the network you will create is on the east coast of the U.S., there is no need to view the entire map. To zoom in on the east coast:

- 1 Click the **Zoom** toolbar button.



- 2 Select the top left corner of the area to zoom in on and drag a rectangle to bottom right corner. Be certain that both Atlanta and Boston are included in the zoomed view.

Using the Zoom Button



Tip: If you are not happy with your first zoom, you can click the **Zoom to Previous** toolbar button, next to the **Zoom** toolbar button, and then zoom in again.

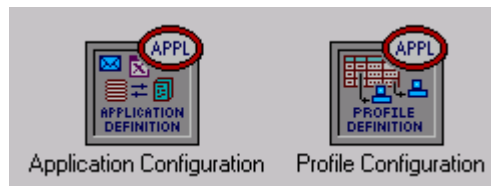
- 3 Continue to zoom until the names of cities, including Boston and Atlanta, appear on the map.

Configuring Applications

It is a good idea to define the profiles and applications that will be used by the LAN before you begin constructing the network. You define the profiles in the profile definition object and applications in the application definition object.

- **A profile** is applied to a workstation, server, or LAN. It specifies the applications used by a particular group of users. You might have one profile for Marketing (heavy use of email; light use of file transfer) and another profile for Engineering (light use of email; heavy use of file transfer).
- **An application** may be any of the common applications (email, file transfer) or a custom application you define. Eight common (“standard”) applications are already defined: Database Access, Email, File Transfer, File Print, Telnet Session, Video conferencing, Voice over IP Call, and Web Browsing.

Profile Definition and Application Definition Objects








To Configure the Application Configuration Object

Follow these steps.

- 1 Open the object palette if it is not already open.
- 2 Drag an **Application Config** object to the project workspace.
- 3 Right-click on the new object and select **Edit Attributes**.
 ➔ The Attributes dialog box opens.
- 4 Click on the question mark next to the **name** attribute to see a description of the attribute. Close the attribute description dialog box when done.

Application Definitions Set to Default

Click here to see a description

Attribute	Value
 name	node 0
 model	Application Con
 + ACE Tier Information	None
 + Application Definitions	None
 + Voice Encoder Schemes	All Schemes

- 5 Set the **name** attribute to **Application Configuration**.

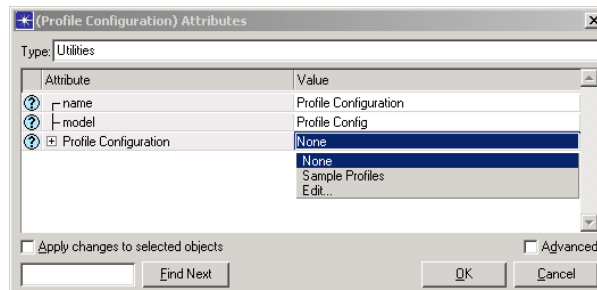
- 6 Change the **Application Definitions** attribute to **Default** by clicking in the attribute's Value column and selecting **Default** from the drop-down list.
 - Selecting **Default** configures the application definition object to make 16 pre-configured applications available for use. These applications are different configurations of the eight standard applications mentioned earlier. Now you will be able to include these applications in the profile you are about to create.
- 7 Click **OK** to accept the changes and close the **Attributes** dialog box.

To Configure the Profile Configuration Object

Follow these steps.

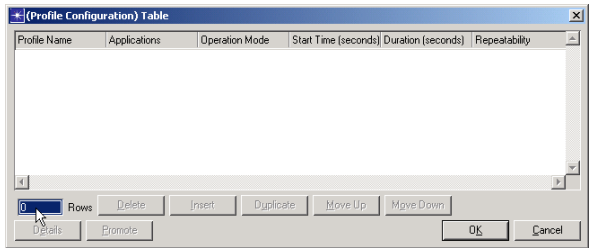
- 1 Drag a **Profile Config** object from the object palette to the project workspace.
- 2 Right-click on the object and select **Edit Attributes**.
- 3 Set the **name** attribute to **Profile Configuration**.
- 4 Change the **Profile Configuration** attribute by clicking in its Value column and selecting Edit... from the drop-down list.

Selecting Edit... from the Menu



➔ The Profile Configuration dialog box appears.

Profile Configuration Dialog Box

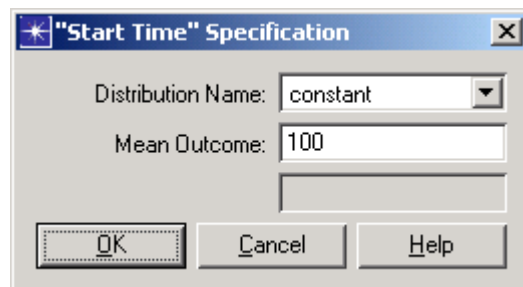


To Define a New Profile and Add It to the Table

Follow these steps.

- 1 Change the number of rows to **1**.
- 2 Name the new profile **LAN Client** by clicking in the Profile Name column of the first row.
- 3 Skip the next column, Application. We'll configure the other attributes first and come back to this one later.
- 4 Change the **Operation Mode** to **Simultaneous**.
- 5 Click in the profile's **Start Time (seconds)** cell to open the Start Time Specification dialog box.
- 6 Select **constant** from the **Distribution Name** pull-down menu.

Start Time Specification Dialog Box



- 7 Set **Mean Outcome** to 100, then click **OK** to close the Start Time Specification dialog box.

- ➔ The Start Time attribute has a value of **constant (100)**.

Do not close the profile configuration attribute dialog box.

You will be modeling FTP performance. That application should be included in the profile.

- 1 Click in the Applications column and choose **Edit...** from the pop-up menu.

- ➔ The **Applications** dialog box appears.

- 2 Change the number of rows to **1**.

- 3 Set the **Name** to **File Transfer (Heavy)** by clicking in the cell and selecting the application from the drop-down list.

The contents of the drop-down list are controlled by the Applications Configuration object. When you selected **Default** as the value for the **Application Definitions** attribute in this object, you enabled this list of applications.

Note that the list includes 16 entries, a heavy and light version for each of the eight standard applications.

4 Set the Start Time Offset to uniform (0, 300).

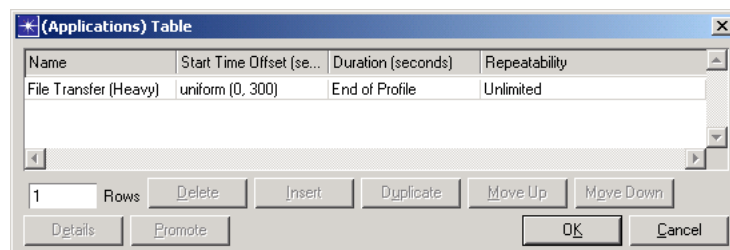
4.1 Verify that the Distribution Name is uniform.

4.2 Set the Minimum Outcome to 0.

4.3 Set the Maximum Outcome to 300, then click OK.

5 Verify that the completed dialog box looks like this:

Completed Applications Dialog Box



- 6 Click **OK** to close the Applications Table dialog box.
- 7 Click **OK** to close the Profile Configuration Table, then click **OK** again to close the Attributes dialog box.

Building the Network

Now that you have set up the scenario, configured the application, and created a profile that uses the application, you are ready to begin constructing the WAN. Because the network contains four identical subnets, you can create the first subnet in Atlanta, with its nodes inside it, and then copy the subnet to Boston, New York, and Philadelphia. You will also copy it to Washington, D.C. and modify it further. This topic focuses on:

- Building subnets
- Copying and pasting network objects
- Modifying subnets
- Connecting subnets

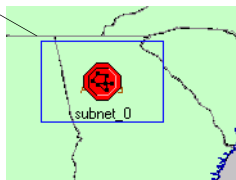
A **subnet** is a single network object that contains other network objects (links, nodes, and other subnets). Subnetworks allow you to simplify the display of a complex network through abstraction.

Subnets help you logically organize your network model. You can nest subnets within subnets to an unlimited degree. For this lesson, you can use subnets as logical containers for the offices in each of the cities. To create a subnet:

- 1 Open the object palette if it is not already open, and move it to the lower right corner of the screen so that it is out of the way.
- 2 Place a subnet over Atlanta.
 - 2.1 Click the subnet icon in the object palette and drag it to the workspace.
 - 2.2 Right-click to turn off node creation.
- 3 Modify the subnet extent of the Atlanta subnet. The subnet extent is the geographic area covered by the subnet, which may be much larger than the actual area you wish to model.

Subnet Extent

Subnet Extent

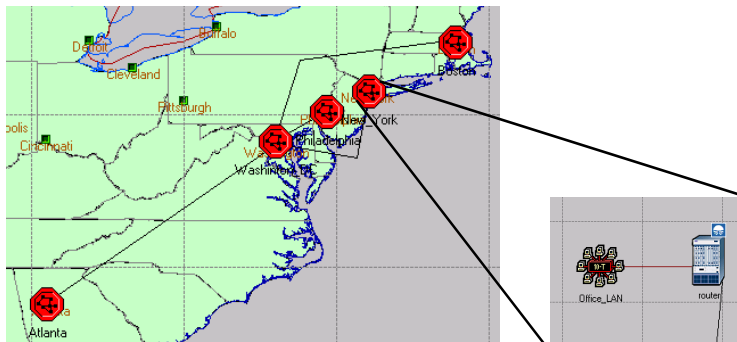


- 3.1 Right-click and select **Advanced Edit Attributes**.
- 3.2 Set the name attribute to Atlanta.
- 3.3 Change the **x span** and **y span** attributes to **.25**. The unit of measure of these attributes is determined by the unit of measure of the top-level area, degrees in this case. The area covered by .25 is the area covered by one-quarter of one degree of latitude or longitude.
- 3.4 Click **OK**. Notice that the subnet extent is now much smaller (it may be hidden underneath the subnet icon).

Double-clicking on a subnet object allows you to see what is “inside” the subnet.

When you double-click on a subnet, IT Guru changes the view to show you what is inside that subnet. Subnets can contain nodes, links, and other subnets. An example topology follows. You will not see this topology right now. You are about to create it.

A Subnet Hierarchy

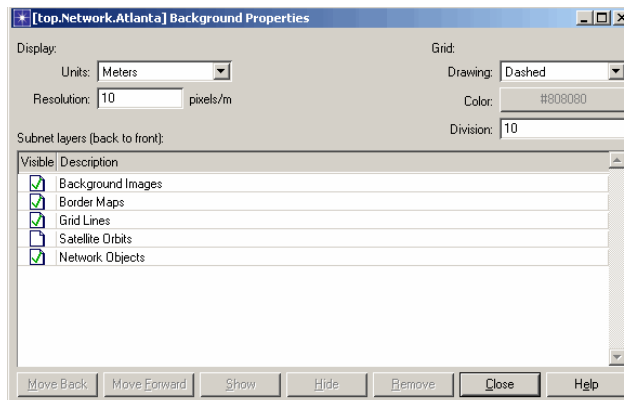


Initially, a subnet's grid properties are based on its parent subnet. If the subnet's grid settings are inappropriate, you can change them to fit your network. To change the grid inside a subnet:

- 1 **Double-click** on the Atlanta subnet.
- 2 Select **View > Background > Set Properties.**
 - ➔ Note that the display grid is in degrees, which is not appropriate for an office.
- 3 Set units to **Meters.**
- 4 Set resolution to **10 pixels/m.**
- 5 Uncheck the Visible checkbox for Satellite orbits.

- 6 Verify that Drawing is set to **Dashed**.
- 7 Set division to **10**.

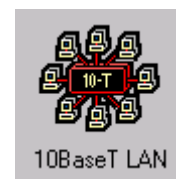
Set View Properties Dialog Box



- 8 Click the **Close** button.

Modeling the East Coast company's network does not require modeling the precise nature of each node in each subnet, so you can represent the subnets with a LAN model. To create a LAN model:

- 1 Place a **10BaseT_LAN** in the workspace.
- 2 Right-click on the **10BaseT_LAN** and choose the **Edit Attributes** menu item.

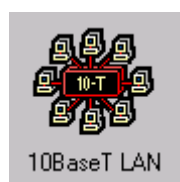


You can change a LAN model's attributes so that it represents a network with a certain number of workstations and a particular traffic profile. To represent one of the East Coast company's satellite offices:

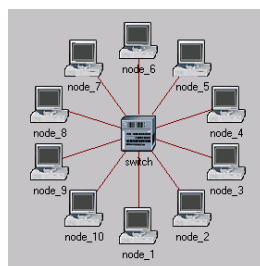
- 1 Change the LAN model's **name** attribute to **Office_LAN**.
- 2 Expand the Applications attribute group.
- 3 Choose **Edit...** for the **Application: Supported Profiles** attribute.
- 4 Change the number of rows to **1**.
- 5 Change the **Profile Name** to **LAN Client**, then click **OK**.
 - ➔ Now this LAN will use the LAN Client profile you created earlier. This profile includes the File Transfer (Heavy) application. The LAN will receive and send traffic that models heavy FTP use.
- 6 Expand the LAN attribute group.
- 7 Verify that the **Number of Workstations** attribute is set to **10**, then click **OK**.

You have now modeled a 10 workstation LAN inside the Atlanta subnet. This single object is equivalent to a 10-workstation star topology LAN.

A Single LAN Object is Equivalent to a LAN Composed of Many Objects



A LAN object



A 10BaseT LAN
with 10 workstations

Because this LAN model is composed of workstations and links only, it must be connected to a router. The router can then be connected to other routers in the network.

To create a router:

- 1 Drag a **BN_BLN_4s_e4_f_sl8_tr4** node (a Bay Networks router) from the object palette to the workspace near the **Office_LAN** node.
- 2 Name the new node **router**.

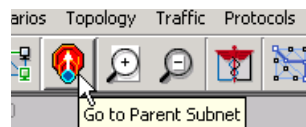
- 3 Connect the **router** and the **Office_LAN** nodes with a **10BaseT** link. Right-click to turn off link creation.

The Atlanta subnet is now configured. Because the subnets in the other cities are identical, you can copy the Atlanta subnet and place it appropriately.

When several subnets or network objects have an identical configuration, you can copy and paste these objects.

To copy the subnet:

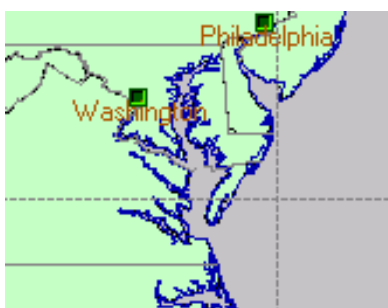
- 1 Return to the parent subnet view by clicking on the **Go to Parent Subnet** button (you can also right-click on the workspace to bring up the workspace pop-up menu, then choose **Go to Parent Subnet** from the menu).
- 2 Select the subnet.
- 3 Select **Edit > Copy** or press **<Control>+c**.



Paste the subnet to each of the four different cities.

- 1 Select **Edit > Paste** or press **<Control>+v** and click on the Washington, DC icon.

Pasting the Subnet

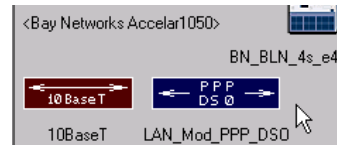


➔ A new subnet appears.

- 2 Press **<Control>+v** again to paste subnets over Philadelphia, New York, and Boston.
- 3 Right-click on each subnet and select Set Name to rename each city's subnet as follows:
 - Washington, D.C.: **Washington_DC**
 - Philadelphia: **Philadelphia**
 - New York: **New_York**
 - Boston: **Boston**

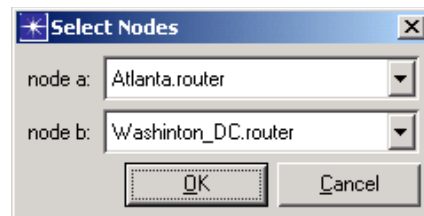
Connect each subnet to the Washington_DC office:

- 1 Select the **LAN_Mod_PPP_DS0** link in the object palette.



- 2 Draw a **LAN_Mod_PPP_DS0** link from **Atlanta** to **Washington_DC**.
 - ➔ A Select Nodes dialog box appears asking which nodes in each subnet are to be endpoints of the link.
- 3 For **node a**, choose the **Atlanta.router** node.
- 4 For **node b**, choose the **Washington_DC.router** node.

Select Nodes Dialog Box



- 5 Click **OK** to establish the link.

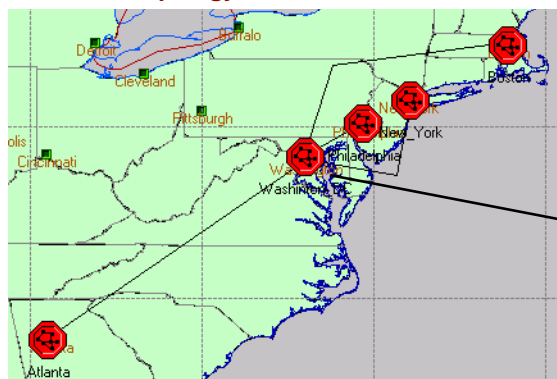
- 6 Repeat this process, drawing links from each city to **Washington_DC**, specifying each city's router as the links' endpoints.

To prevent overlapping links, you can click on intermediate points in the workspace to make a link follow a path before finally clicking on its destination node.

- 7 Right-click to turn off link creation.

The network should resemble the following:

The Initial Topology



Draw links from each city to Washington_DC. Make sure the links do not overlap.

To complete the network, the main office in Washington D.C. needs to have a switch and a server added to it. To configure the network in D.C.:

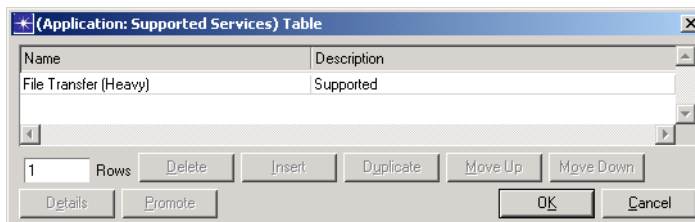
- 1 Double-click on the **Washington_DC** subnet to enter its subnet view.
- 2 Place one **<Bay Network Accelar1050>** switch and one **ethernet_server** node in the workspace.
- 3 Rename the **<Bay Network Accelar1050>** node to **switch**.
- 4 Rename the **ethernet_server** to **FTP**.
- 5 Connect the “router” and the “FTP” nodes to the switch with **10BaseT** links. Right-click to turn off link creation.
- 6 Close the object palette.

Configure the Server to Support the FTP Application

Follow these steps.

- 1 Open the Attributes dialog box for the **FTP** server.
- 2 Expand the Applications attribute group.
- 3 Choose **Edit...** for the **Application: Supported Services** attribute.
- 4 Change the number of rows to **1**.
- 5 Select **File Transfer (Heavy)** from the **Name** column pop-up menu.

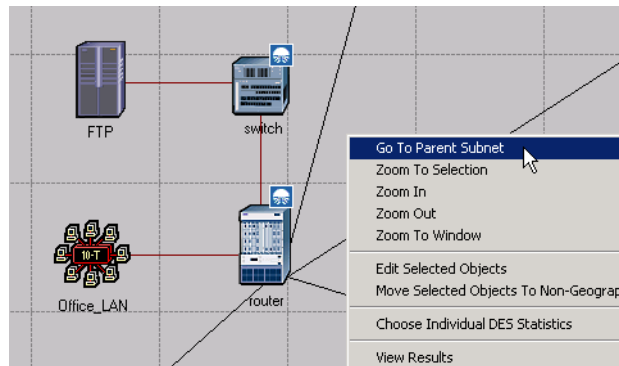
Configure the Server to Support the Application



- 6 Click **OK** to close the Supported Services dialog box, and then click **OK** to close the (FTP) Attributes dialog box.

The Washington, D.C. subnet is now complete and should resemble the following figure.

Washington, D.C. Subnet



- 7 Use the workspace pop-up menu to return to the parent subnet view.
- 8 Save the project by selecting **File > Save**.

Background Load

Now that you have created a model to act as a baseline for the performance of the East Coast company's network, you can add background traffic to the links connecting the cities and compare the results from the two scenarios.

This topic focuses on

- Duplicating a scenario
- Implementing a varying background load on the links

Background loading is an efficient means of modeling a known traffic load on a link.

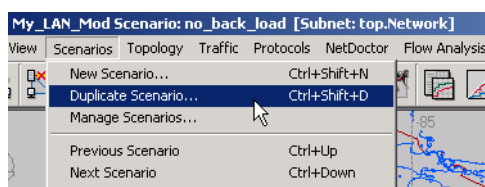
Network studies show that traffic rises gradually over the course of the day as employees arrive and begin work. You can use background link utilization to model this pattern.

Because you want to compare the performance of the network with and without background utilization, you need to prepare two scenarios, one for each situation.

You will duplicate the existing scenario, then add background traffic to it:

1 Select **Scenarios > Duplicate Scenario...**

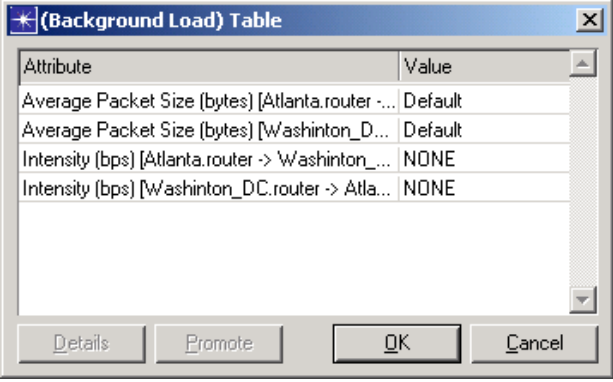
Scenarios Menu



2 Name the scenario **back_load**. and click **OK**.

Background Load is an attribute of each link. To set background load on the links between the cities:

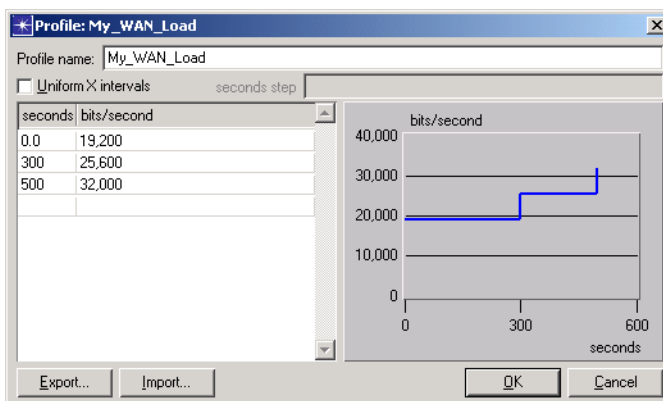
- 1 Select the links between subnets. Right-click on the link between Washington, D.C. and Atlanta, then choose **Select Similar Links** from the pop-up menu.
- 2 Display the **Attributes** dialog box for the link between Washington, D.C. and Atlanta.
- 3 Click in the Value cell for the **Background Load** attribute and select **Edit...** from the pop-up menu.
 - ➔ A **Background Load** table appears.

Background Load Table

Attribute	Value
Average Packet Size (bytes) [Atlanta.router -> Washinton_DC.router]	Default
Average Packet Size (bytes) [Washinton_DC.router -> Atlanta.router]	Default
Intensity (bps) [Atlanta.router -> Washinton_DC.router]	NONE
Intensity (bps) [Washinton_DC.router -> Atlanta.router]	NONE

- 4 Click on the Value cell for “Intensity (bps) [Atlanta.router -> Washington_DC.router]” and select Edit... from the pop-up menu.
 - ➔ The **Traffic Intensity (A -> B) Attribute Profile** dialog box appears.
- 5 Set the Profile name to “My_WAN_Load”.
- 6 Complete the Profile table as shown in the following figure.

Profile Table



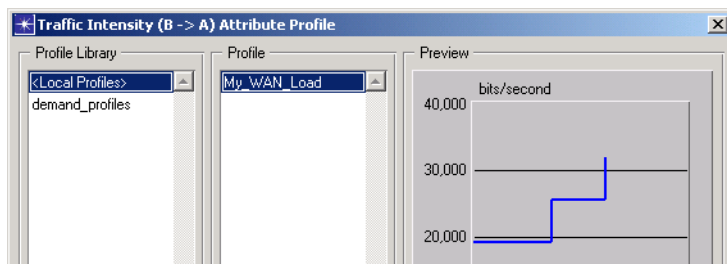
The intensity table above indicates that for the first 300 seconds of the simulation, the background load on this link will be 19,200 bps (that is, 30 percent of the DS0 capacity of the link). For the next 200 seconds, the background load on the link will be 25,600 bps (about 40 percent of the link capacity), and for the last part of the simulation, the load will be 32,000 bps (about 50 percent of the link capacity).

- 7 Click **OK** to close the profile table.

- Click in the Value cell of the Intensity (bps) [Washington_DC.router -> Atlanta.router] attribute and choose **Select...** from the pop-up list.

➔ The following table appears.

Attribute Profile Table



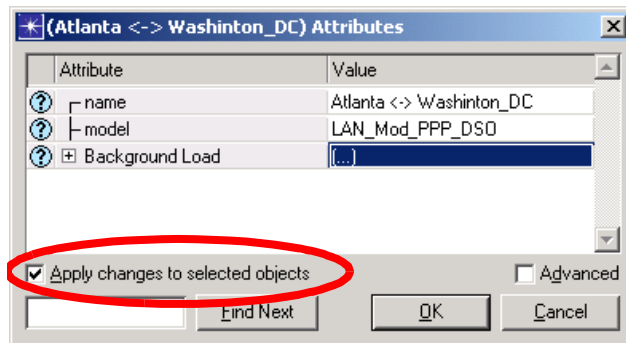
- Select <Local Profiles> from the Profile Library list and "My_WAN_Load" from the Profile list.
- Click **OK** to close the **Profile Selection** dialog box.
- Click **OK** to close the **Background Load** table dialog box.

Do not close the link attributes dialog box.

The last step in setting background load is to apply the changes made to the Atlanta–Washington, D.C. link to all selected links.

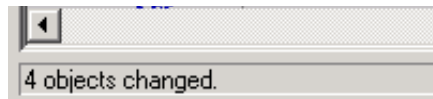
- 1 Check the **Apply changes to selected objects** check box in the **Atlanta <-> Washinton_DC** Attributes dialog box.

Link Attributes Dialog Box



- 2 Click **OK** to close the Attributes dialog box, then click **Yes** if you are prompted to continue.
➔ Note that “**4 objects changed.**” appears in the message area.

Confirming Message



- 3 Save the project by selecting **File > Save**.

Collecting Statistics

Now that you have configured both scenarios (one without background load as a baseline, and one with background load), you are ready to collect data and analyze it.

The relevant statistics for this network are the throughput and utilization statistics for the links and the global FTP download time for the network.

This topic focuses on:

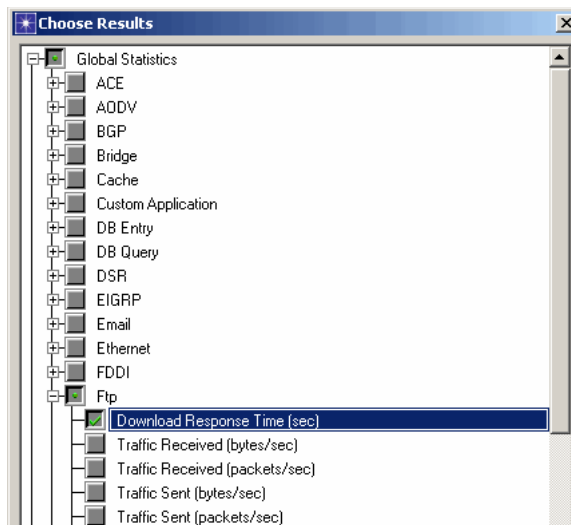
- Specifying statistics to collect in each scenario
- Switching between scenarios
- Running multiple simulations

To Collect Statistics in the back_load Scenario

Follow these steps.

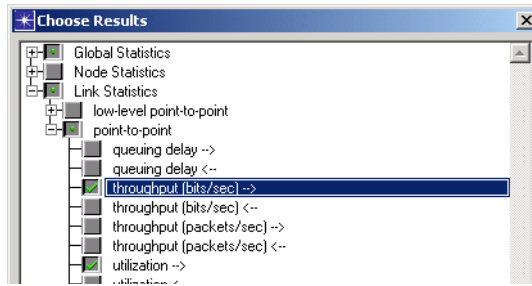
- 1 Right-click in the workspace to display the workspace pop-up menu, and select **Choose Individual DES Statistics**.
- 2 Select the **Global Statistics > Ftp > Download Response Time (sec)** statistic.

Selecting the FTP Download Response Time Statistic



- 3 Select the **Link Statistics > point-to-point > throughput (bits/sec) -->** and **utilization -->** statistic.

Selecting the Throughput and Utilization Statistics

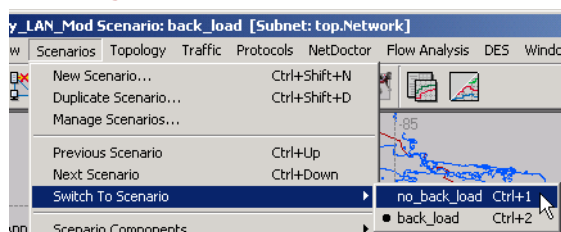


- 4 Click **OK** to close the **Choose Results** dialog box.

The throughput and utilization statistics that you have specified will be collected for every link in the simulation. In order to compare the statistics in the **back_load** scenario to the **no_back_load** scenario, the same statistics must be collected in the **no_back_load** scenario. To change scenarios and collect statistics:

- 1 Select **Scenarios > Switch To Scenario**, then choose **no_back_load**.

Selecting the no_back_load Scenario



- 2 Collect the same statistics that you did in the **back_load** scenario:
 - Global Statistics > Ftp > Download Response Time (sec)
 - Link Statistics > point-to-point > throughput (bits/sec)-->
 - Link Statistics > point-to-point > utilization -->
- 3 Close the **Choose Results** dialog box and save the project.

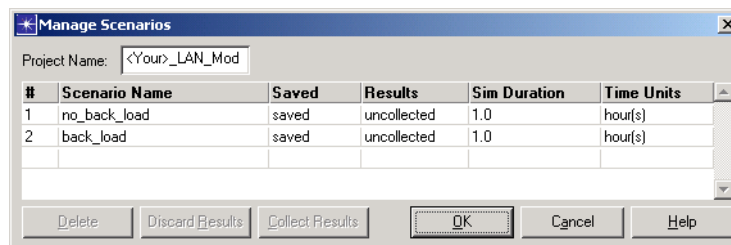
You are now ready to run the simulations to collect the statistics you have specified.

Using the **Manage Scenarios** dialog box, you can rename scenarios, change their order, and run single or multiple simulations.

Instead of running each simulation separately, you can batch them together to run consecutively. To run multiple simulations:

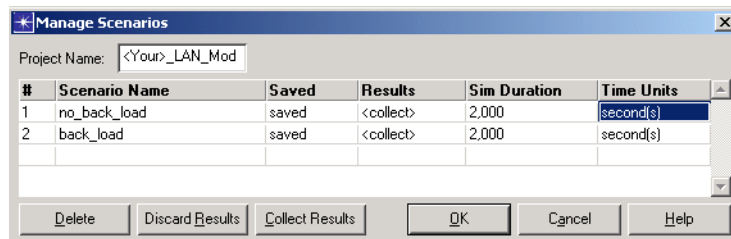
- 1 Select **Scenarios > Manage Scenarios...**
 ➔ The Manage Scenarios dialog box appears.

Manage Scenarios Dialog Box



- 2 Click on the **Results** value for the **no_back_load** and **back_load** scenarios and change the value to **<collect>**.
- 3 Set the **Sim Duration** value for each scenario to **2,000** and the **Time Units** to **seconds**.

Running the Simulation from the Manage Scenarios Dialog Box



4 Click OK.

- This runs simulations for both scenarios. A Simulation Sequence dialog box shows the simulation progress. Close the dialog box when the simulations are done.
- If your simulation does not complete, if no results were collected, or if the results vary significantly from those shown, you will have to troubleshoot your simulation. See *"Troubleshooting IT Guru Tutorials"*.

Comparing Results

You are now ready to examine the results of the two scenarios. Because you collected the same statistics in each scenario, you can use the Compare Results feature to look at them together.

To view the results from two or more different scenarios against each other, you can use the **Compare Results** feature. You can also apply different built-in filters to the graphs.

This topic focuses on:

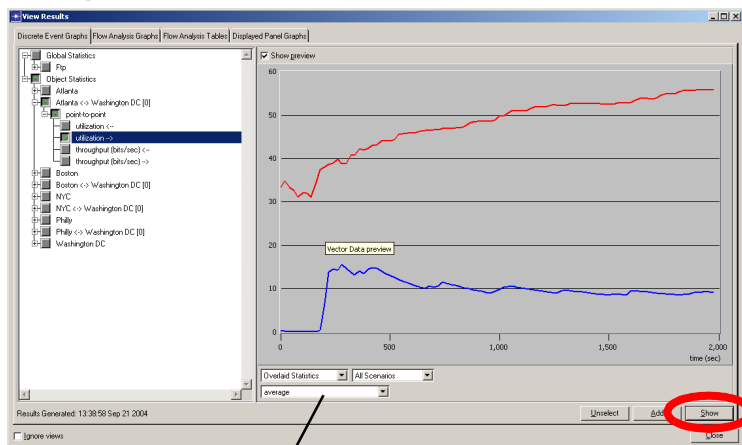
- Comparing results between scenarios
- Applying filters to graphs

The **Compare Results** feature shows results from two or more scenarios. To look at link utilization:

- 1 Display the workspace pop-up menu and choose **Compare Results**.
- 2 In the Compare Results dialog box, select **Object Statistics > Network > Atlanta <-> Washington_DC [0] > point-to-point > utilization -->**.

- 3 Because utilization varies over the course of a simulation, it is helpful to look at the time average for this statistic. Change the Filter menu from **As Is** to **average**.

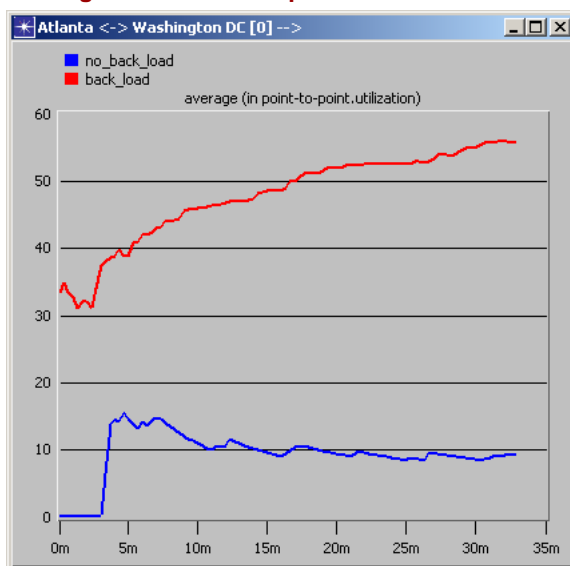
Setting the Filter



Filter menu

- 4 Click **Show** to display the graph. Your graph should resemble the following figure, though it will not match exactly:

Average Utilization Compared



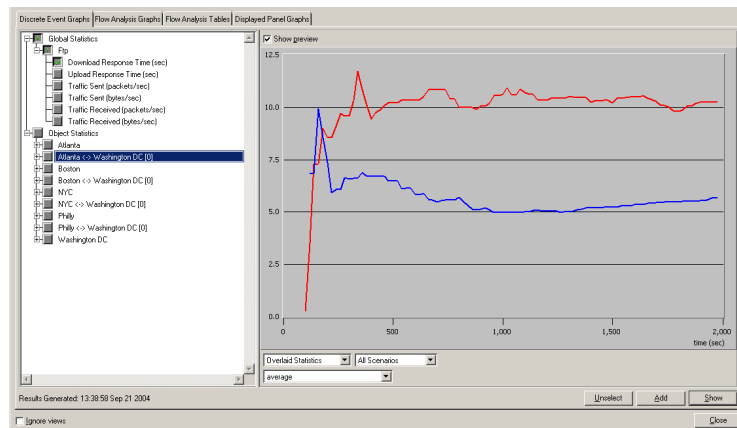
The network with the background load (back_load) shows higher utilization. You may want to look at the utilization of other links to determine the maximum utilization of any link.

You can also compare the link throughputs (bps) by selecting **Object Statistics > Network > Atlanta <-> Washington_DC [0] > point-to-point > throughput (bits/sec) -->**.

Next, let's look at Global FTP response time:

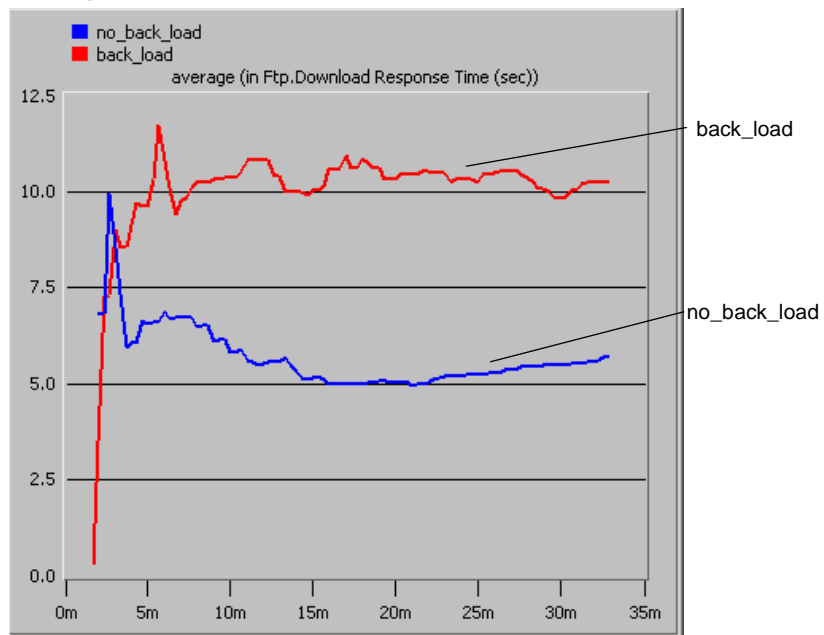
- 1 Move the link utilization graph aside and click the **Unselect** button in the Compare Results dialog box.
- 2 Check the **Global Statistics > Ftp > Download Response Time (sec)** statistic in the Compare Statistics dialog box.
- 3 Verify that the Filter menu shows **average**, then click **Show**.

Select Average from the Filter Menu



The graph should resemble the following figure:

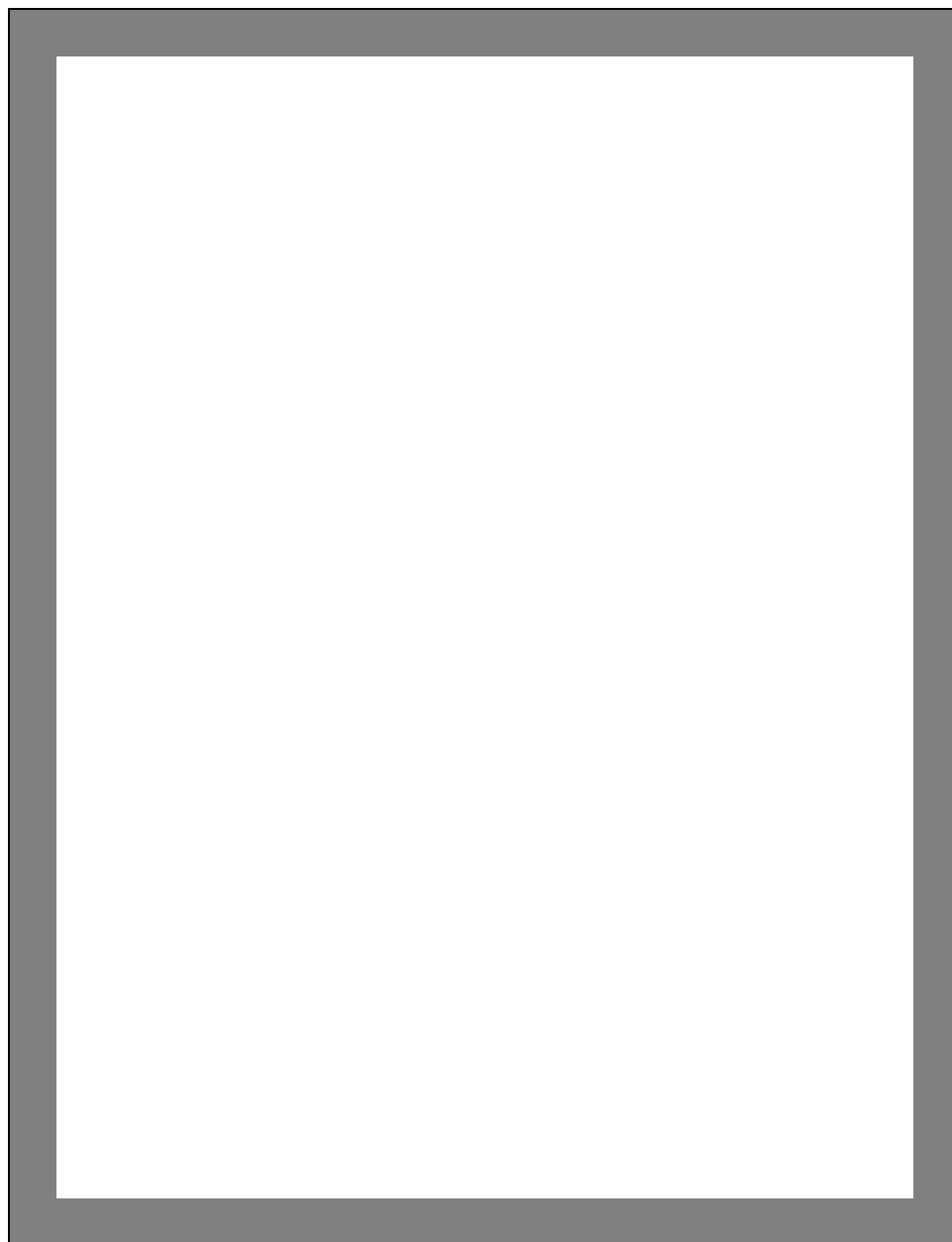
Average Download Response Time Compared



This graph shows that, as steady state is reached, response time increases by several seconds when the link is heavily loaded.

- 1 Select **File > Close** and save changes before closing.

Now that you have completed the LAN lesson, you can move on to the Web Reporting lesson. This lesson presents 's web reporting feature and how it can help you to make informed decisions about a network's behavior. Return to the main tutorial menu and choose **Web Reporting** from the list of available lessons.



4 Web Reporting

Web Reporting

Overview

This lesson focuses on using web reports to view and analyze the results of your simulations.

You will learn how to

- Choose a report
- Generate a web report
- Use the information provided in the report to make informed decisions about a network's behavior

Web reports allow you to look at the results of your simulations in several different ways and to make those results available to many people. You can begin with a high-level summary of the results, then use hyperlinks within the reports to focus on problem areas in a network. In this lesson, we will create a web report and use it to analyze the performance of two different traffic contracts (ABR and CBR) on a small ATM network.

Getting Started

Because this lesson focuses on web reports, you will use an existing project instead of creating a new network model. To load the project

- 1 Start IT Guru, if it is not already running.
- 2 Select **File > Open...**, and select **Project** from the pull-down menu.
- 3 Navigate the tree to the directory where the required tutorial resides. The default location is:
`<opnet_dir>\models\std\tutorial_req\basic`
- 4 Select **web_report** from the list of available projects, and click **Open**.
 - ➔ The **web_report** project (an ATM network) opens in the Project Editor.
- 5 Select **File > Save As...**, rename the project **<initials>_web_report**, then save it in your `op_models` directory.

The ATM network used in this lesson is designed to show the differences in performance between two types of ATM traffic contracts, CBR (constant bit rate) and ABR (available bit rate).

In the CBR scheme, the traffic contract guarantees a certain amount of bandwidth at all times for each user. The advantage to this scheme is that each user gets a pre-defined amount of bandwidth, regardless of other traffic on the link. However, this limits the total number of users on a link at any one time. This scheme is useful when a high quality of service is required, as with voice or video traffic.

In the ABR scheme, the traffic contract does not guarantee a certain bandwidth; traffic uses whatever is left after the CBR traffic is handled. The advantage to this is that a large number of users can access the same link at the same time. However, the available bandwidth is divided among those users, possibly increasing response time. This scheme is useful when a high quality of service is not necessary, as with FTP traffic.

Specifying and Generating a Web Report

The ATM network used for this lesson consists of

- an FTP client using the ABR scheme
- a voice client using the CBR scheme
- 2 switches
- an FTP server
- a voice server

You can view the attributes for the client and server objects for more details on the setup of the network. For this network we are interested in the following statistics:

- Client Traffic Sent
- Client FTP Response Time
- Voice Application Packet End-to-End Delay

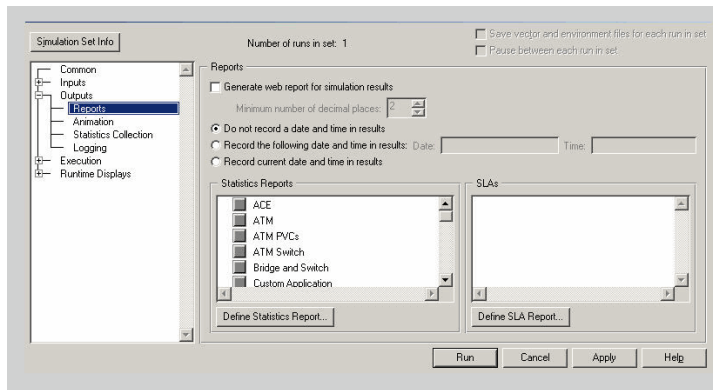
Instead of selecting **Choose Individual Statistics** to collect these statistics, you can choose them as part of a report whose statistics are collected in report form during the simulation. You can choose a report before you run a simulation:

- 1 Click the **Configure/Run Discrete Event Simulation (DES)** toolbar button.



- 2 Select **Outputs > Reports** from the treeview.

“Reports” Page in Configure/Run Simulation Dialog Box



- 3 In the **Statistics Reports** pane, select **FTP&VOICE**.

You can also define your own report that contains any of the available statistics.

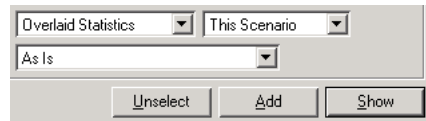
Next, you can run the simulation:

- 1 Select **Common** in the treeview.
 - Note that the simulation is set to run for 1,000 seconds of simulation time.
- 2 Click the **Run** button.
 - The simulation should take about two minutes to run, depending on the speed of your machine.
- 3 When the simulation completes, close the Simulation Sequence dialog box.

Before generating a web report, you can include some graphs that would not normally appear in a report (for example, graphs that show results stacked or overlaid). To include graphs, first display them in the Project Editor, then generate the report with the “Include graphed results” option selected in the **Specify Report** dialog box:

- 1 Right-click anywhere in the workspace, and select **View Results** from the pop-up menu.
- 2 Expand the **Report FTP&VOICE** hierarchy.
➔ The hierarchical view appears.
- 3 Select the following two statistics under **Report FTP&VOICE > Object Statistics**:
 - **stn_abr_ftp_client > Client Ftp > Upload Response Time (sec) <FTP/File Transfer (heavy)>**
 - **stn_cbr_voice_client > Voice Application > Packet End-to-End Delay (sec)**

- 4 Set the view (left pull-down menu) for **Overlaid Statistics**. Verify that the filter (bottom pull-down menu) is set to **As Is**, then click **Show**.



➔ The graph opens in the workspace.

The significance of this graph is discussed later. For now, you are ready to generate and view the web report.

- 5 Click Close in the **View Results** dialog box, but leave the graph window open.

There are two steps to creating a web report: generating it and viewing it in a web browser. To generate the report

- 1 Select **DES > Results > Generate Web Report...**

➔ The **Specify Report** dialog box displays.

2 Click **OK** to generate a report with the default name.

- A web report is generated and placed in the directory specified by the **web_rep_storage_dir** preference. If **web_rep_storage_dir** is unspecified, the web report will be placed in **<user_home_dir>\op_admin\tmp**.

To view the report

1 Select **DES > Results > Launch Last Web Report**.

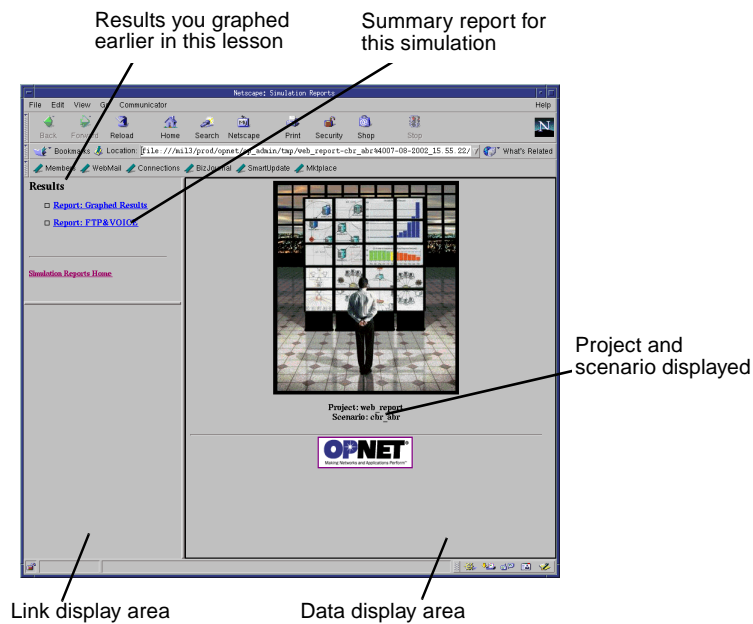
- IT Guru launches the web browser specified by the **browser_prog** preference and opens the report index page.

If the browser does not open, check that the **browser_prog** preference specifies the correct path. You can also open the browser manually and point to the **index.html** file in the directory specified by the **web_rep_storage_dir** preference.

General Report Format

When a web report opens, the index page is displayed. The index page has several links, as shown in the following figure.

Index Page of a Web Report



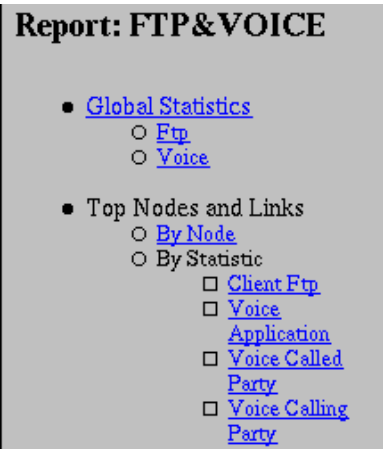
Results by Node and Statistic

Web reports present data about a simulation in several different kinds of reports.

Let's look at the FTP & Voice report:

- 1 In the Results area, click on the **Report: FTP&VOICE** link.
 - ➔ Links to information collected in the Custom Application report appear in the link display area.

Link Display Area of a Web Report



Report: FTP&VOICE

- [Global Statistics](#)
 - [Ftp](#)
 - [Voice](#)
- Top Nodes and Links
 - [By Node](#)
 - By Statistic
 - [Client Ftp](#)
 - [Voice](#)
 - [Application](#)
 - [Voice Called Party](#)
 - [Voice Calling Party](#)

- 2 First, let's look at the global statistics collected. Under the **Global Statistics** link, click on **Voice**.
 - ➔ A table of the Voice global statistics appears.

The main statistics of interest in this table are Traffic Sent and Traffic Received. Note that in this main table, the number of packets/second received network-wide equals the number of packets/second sent, so we know that no traffic is being dropped.

Global Statistics for the Voice Application

Statistic	Average	Maximum	Minimum
Voice Packet Delay Variation	0.0000000000	0.0000000000	0.0000000000
Voice Packet End-to-End Delay (sec)	0.049769	0.049769	0.049764
Voice Traffic Received (bytes/sec)	4,797	16,000	0
Voice Traffic Received (packets/sec)	149.93	500.00	0.00
Voice Traffic Sent (bytes/sec)	4,797	16,000	0
Voice Traffic Sent (packets/sec)	149.92	500.00	0.00

From this table, you can click on the links and view the graphs of each of the global statistics. However, for this particular scenario, you are primarily interested in the traffic sent and received at the ABR FTP node and CBR Voice node, so global statistics are of less interest.

Of particular interest in this report is the Top Nodes and Links section, which provides you with a quick, comprehensive view of which objects in your network may be problems. It lists objects that were most “active” during a simulation. “Active” in this context means that the object generated the highest values for any statistic.

In this network, there were only two nodes for which statistics were collected, the ABR and CBR client nodes. The performance of these nodes can be compared in the tables in the Top Nodes and Links section.

Top Nodes and Links

Report: FTP&VOICE

- [Global Statistics](#)
 - [Ftp](#)
 - [Voice](#)
- Top Nodes and Links
 - [By Node](#)
 - By Statistic
 - [Client Ftp](#)
 - [Voice](#)
 - [Application](#)
 - [Voice Called Party](#)
 - [Voice Calling Party](#)

- 1 In the link display area, click on the **By Node** link, under Top Nodes and Links.

The Top Nodes Summary displays.

Top Nodes Summary Report

Node	Client Ftp Download File Size (bytes)	Client Ftp Download Response Time (sec)	Client Ftp Traffic Received (bytes/sec)	Client Ftp Traffic Received (packets/sec)	Client Ftp Traffic Sent (bytes/sec)	Client Ftp Traffic Sent (packets/sec)
sta_abr_ftp_client	-	-	3.072	0.006	300	0.006
sta_cbr_voice_client	-	-	-	-	-	-
sta_cbr_voice_server	-	-	-	-	-	-

Any object that generates an extreme value for any collected statistic is considered a top node or link. In this example, there are only three objects. Larger networks with more nodes and additional collected statistics could have more entries in the table.

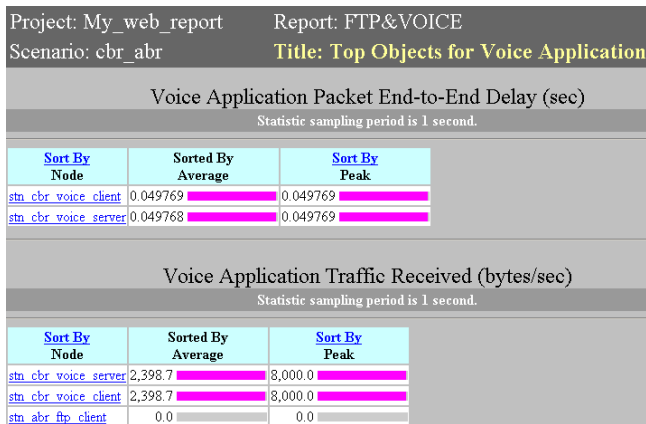
You can specify the number of objects considered “top” by setting the **report_top_n** preference before you generate the report.

From the Top Nodes and Links table, you can see that the overall results are as expected: the ABR client running FTP produced more traffic than the CBR client running voice.

Each of the statistic names in the table is a link to another table that shows each statistic on a per-node basis. Let's take a closer look at the response time:

- 1 Click on the **Voice Application Packet End-to-End Delay (sec)** heading, which is also a link (you might need to scroll to the right to see this column heading).
 - The Top Objects for Voice Application report opens. Similar graphs for the other statistics collected on a per-node basis also appear on this page.

Top Objects for Voice Application Report

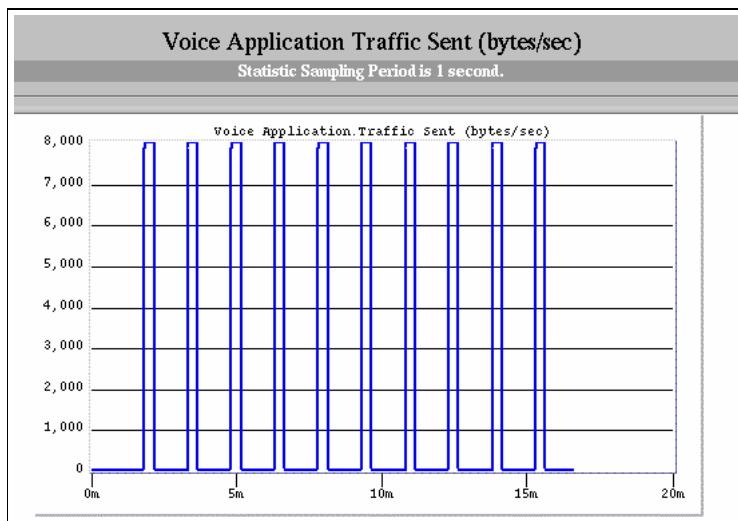


For each statistic discussed, your results may vary slightly, but general trends should be the same.

Click on the headings to sort the table by Node, Average, or Peak. Sorting the table by different measures makes it easy to see which objects have the highest average values versus which objects have the highest peak values for a particular statistic. Let's take a closer look at **stn_cbr_voice_client** in the **Voice Application Traffic Sent (bytes/sec)** table.

- 1 Click on the **stn_cbr_voice_client** link in the **Voice Application Traffic Sent (bytes/sec)** table. (You might need to scroll down to see this table.) The graph is shown in the following figure.

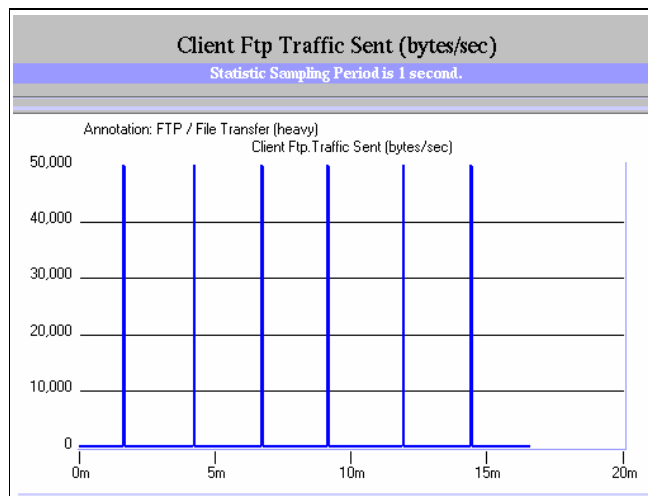
CBR Client Sending Voice Application Traffic



Next, let's look at the traffic received at the ABR client and see how its traffic pattern differs from the CBR client.

- 1 Click the **Back** button in your browser until the Top Nodes Summary page appears.
- 2 Click on the **Client Ftp Traffic Sent (bytes/sec)** heading.
- 3 Click on the **stn_abr_ftp_client** link in the **Client Ftp Traffic Sent (bytes/sec)** table.
➔ The graph should resemble the following figure.

ABR Client Sending FTP traffic



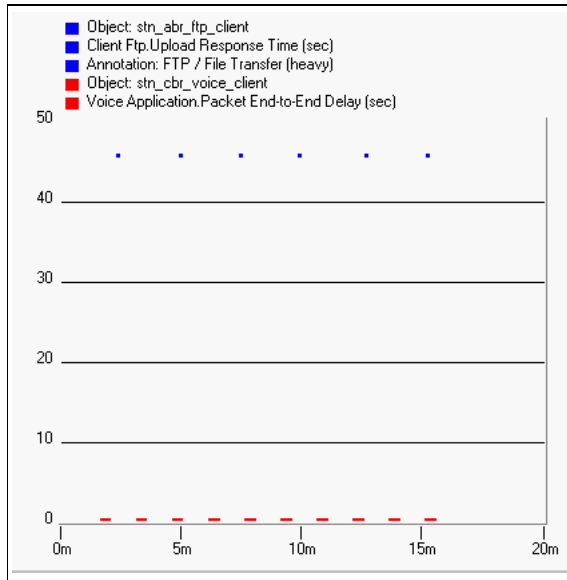
Compare the graph of the ABR client sending FTP traffic with the graph of the CBR client sending voice application traffic. The ABR client, as expected, sends traffic less frequently.

In addition to looking at the tables and graphs generated by the web report, you can view the graphs that were displayed in the Project Editor window when you generated the web report. This can be especially useful when you want to have graphs that compare two or more objects or statistics (rather than comparing them in a table).

These graphs are contained in the **Graphed Results** link in the report. To view this information:

- 1 Click on the **Report: Graphed Results** link.
 - ➔ Links to the graphs that were open in the Project Editor when you generated the web report appear in the link area.
- 2 Click on the link to **FTP&VOICE**.
 - ➔ The graph of response time should resemble the following figure.

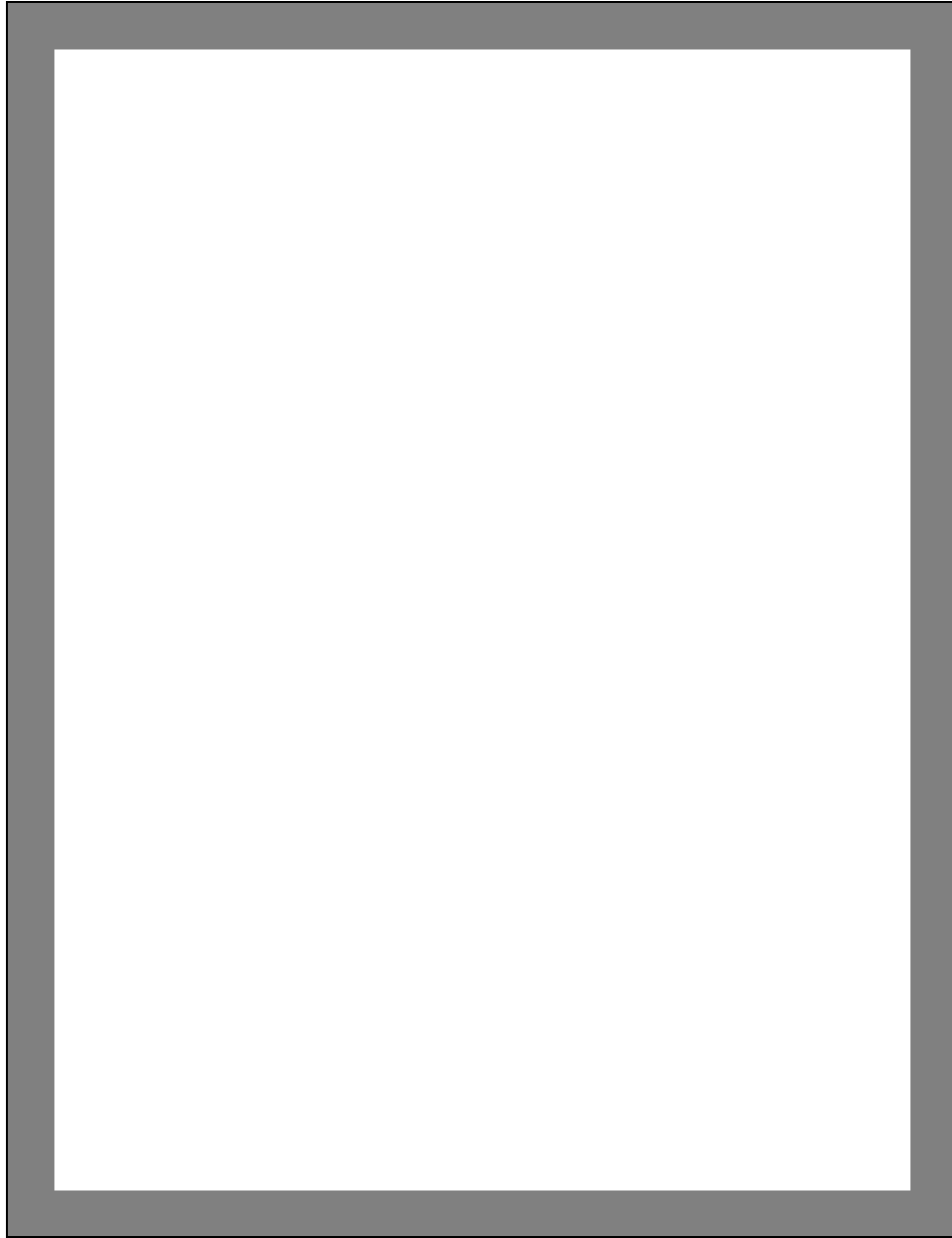
Response Times for CBR and ABR Clients

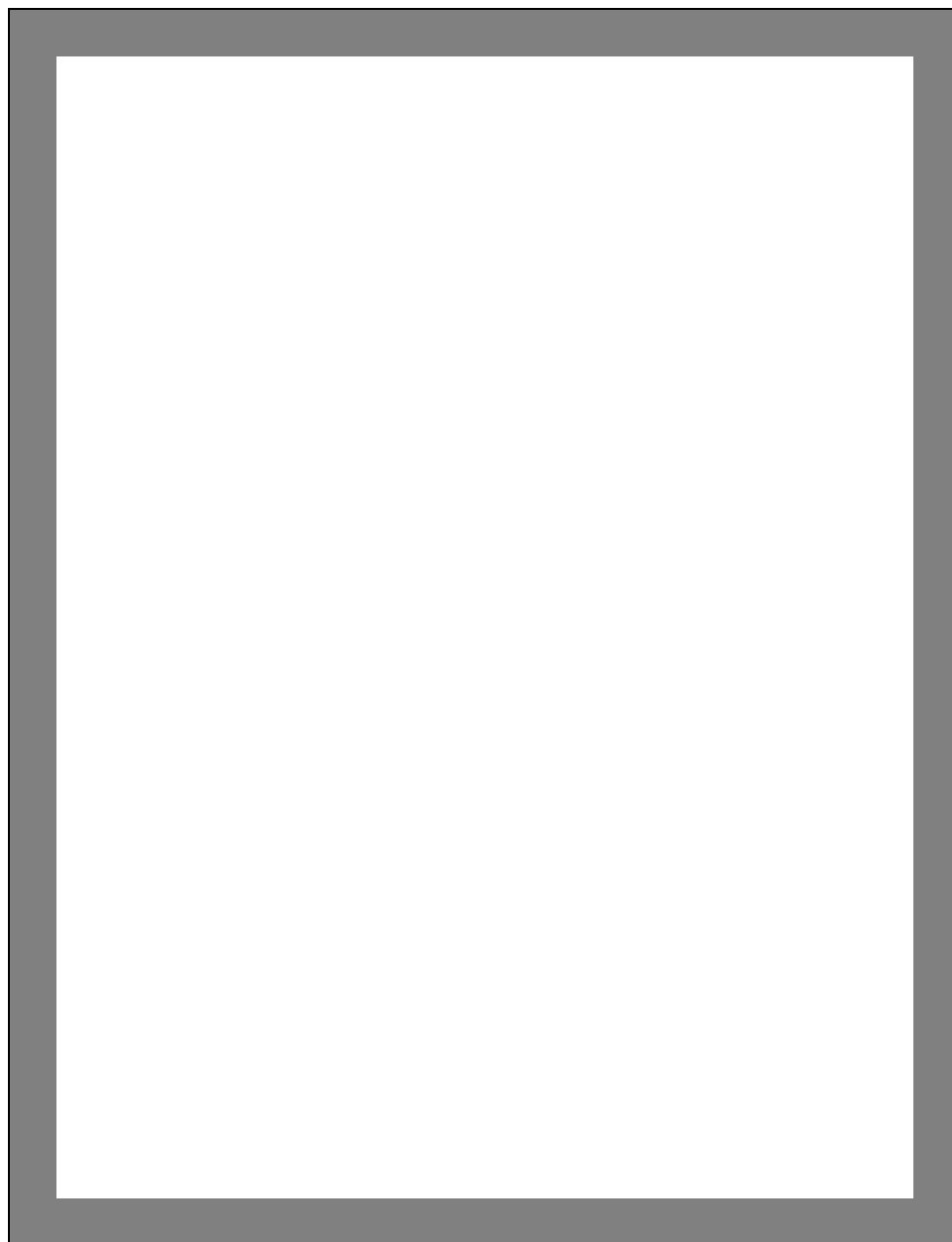


As expected, the response time for the CBR client running the voice application is less than that for the the ABR client running the FTP application.

You have now completed the Web Reporting lesson. If you wish, you can explore the web report further on your own. When you are finished, close the web browser and save your project.

The next lesson, Importing Traffic, illustrates the traffic import capabilities of IT Guru. If you will not be using IT Guru's import capabilities, you might want to skip the Importing Traffic lesson and move on to the ESP lesson. Return to the main tutorial menu, and choose the desired tutorial from the list of available lessons.





5 Importing Traffic

Importing Traffic

Overview

This lesson demonstrates the traffic import capabilities of IT Guru. In this lesson, you will

- Examine a ready-made model that contains explicit traffic
- Import conversation pair traffic
- Run several simulations to see the effect of traffic growth on the network

In earlier tutorials, you built a network manually by dragging objects from a palette, then you added background load traffic by specifying throughput.

This tutorial shows how you can import traffic from external sources. You might import a router configuration file or VNE database to create the topology, then import data from different tools, such as NetScout nGenius, Cisco Netflow, or a custom text file to create the traffic.

Reviewing the Ready-made Model

This tutorial uses a ready-made model with explicit traffic.

- 1 Choose **File > Open**.
 - ➔ The **Open** dialog box appears.
- 2 With **Files of type:** set to **Project**, navigate the treeview in **Model directories:** to the **basic** folder using the following path:
`<opnet_dir>\models\std\tutorial_req\basic`
 - ➔ When you select **basic**, the names of the files in the basic folder appear in the **Files:** pane.
- 3 Select **Imp_Data** from the list of files, then click **Open**.
 - ➔ The network model opens in the workspace.
- 4 Choose **File > Save As**, select your default models directory in the treeview, and save the project as **<your_initials>_Imp_Data**.

There are several types of traffic in IT Guru:

- **Explicitly generated traffic** is user-created; you specify the size of the transactions and the number of transactions per time unit according to a chosen distribution. You create explicit traffic by configuring **Application Definition** and **Profile Definition** objects.
- **Traffic flows** (also called conversation pairs or background routed traffic) are special objects that specify end-to-end traffic between source and destination nodes. You can create traffic flows manually using demand objects from the object palette or import them from an external source (**Traffic > Import Traffic Flows...**).
- **Link Baseline Load traffic** specifies the background traffic in bits per second. You can create this traffic by configuring the **Background Load** attribute of a link or by importing traffic from various sources using **Traffic > Import Baseline Loads**.

In the LAN Modeling tutorial, you used link baseline load (background load) traffic when you created traffic on the links between subnets in the network. In this lesson you will use

- Explicitly-generated traffic to test a “what-if” scenario
- End-to-end traffic flows imported from an ASCII traffic archive

To get values for such statistics as delay and response time on a network with background traffic, you must have explicit traffic.

To create explicit traffic (such as an FTP application), you would normally do the following:

- Add and configure Application Definition and Profile Definition objects
- Choose statistics

To run a baseline simulation, you would also need to configure the simulation.

Because you've done these tasks in earlier tutorials, you do not have to repeat them here. The ready-made network model is already configured correctly.

However, to help you become more familiar with the configuration tasks, we will look briefly at the following:

- Application Definition and Profile Definition objects
- Choose Statistics dialog box
- Configure Discrete Event Simulation dialog box

Review the Application Definition and Profile Configuration objects.

- 1 If you are not already in the **192_9_200** subnet, double-click on it.
- 2 In the Application Definition object's Edit Attributes dialog box, verify that the **Application Definitions** attribute is set to **Default**. Click **OK** to close the dialog box.
- 3 In the Profile Configuration object's Edit Attributes dialog box, open the **Profile Configuration** attribute table by clicking in the **Value** field and choosing **Edit...** from the pull-down menu.
- 4 Verify that the attributes are set as follows:
 - **Profile Name: FTP (Light)**
 - **Applications:** Choose **Edit...** to display the **Applications** table, verify that the **File Transfer (Light)** application is set, then click **OK**.
 - **Operation Mode: Serial (Ordered)**
 - **Start Time (seconds): uniform (100, 110)**
 - **Duration (seconds): End of Simulation**

- **Repeatability: Once at Start Time**

- 5 Click **OK** to close the **Profile Configuration** table, then **OK** to close the **Edit Attributes** dialog box.

Verify that the **FTP (Light)** profile is assigned to the LAN objects and that the application **File Transfer (Light)** is supported on the LAN server.

To do this, you will use the **Select Objects...** command, specifying that the Supported Profile value be **FTP (Light)** and that the Supported Services value be **File Transfer (Light)**. You know that the network contains six LAN objects, so if these attribute values are assigned to every LAN object, the command will report six objects selected.

- 1 Choose **Edit > Select Objects...**
- 2 Configure the dialog box as shown in the following figure **BUT DO NOT CLICK OK.**

Logical Object Selection Dialog Box

Object type to be selected is node

Search scope is current subnet

Proposition	Attribute Name	Condition	Value
Require	Application: Supported Profiles	=	...

Proposition is Require

Attribute Name is Application: Supported Profiles

Click in the Value cell for this row and select Edit...

3 Specify the value for the Application: Supported Profiles attribute.

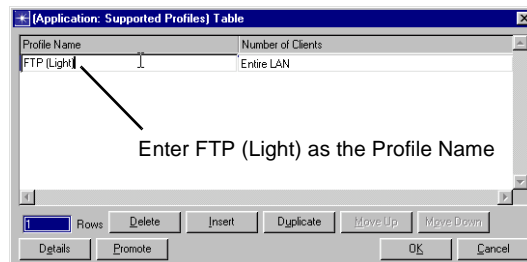
3.1 Click in the Value cell of the first row in the Attribute Criteria table and select Edit...

➔ **The Application: Supported Profiles Table dialog box appears.**

3.2 Click in the **Rows** field and add a row to the table.

3.3 Click on the **None** text string in the **Profile Name** column and type **FTP (Light)**.

Application: Supported Profiles Dialog Box



3.4 Click **OK** to close the **Application: Supported Profiles** dialog box.

4 In the Logical Object Selection dialog box, add a second row to the Attribute Criteria table and configure it as follows:

- **Proposition: Require**
- **Attr. Name: Application: Supported Services**

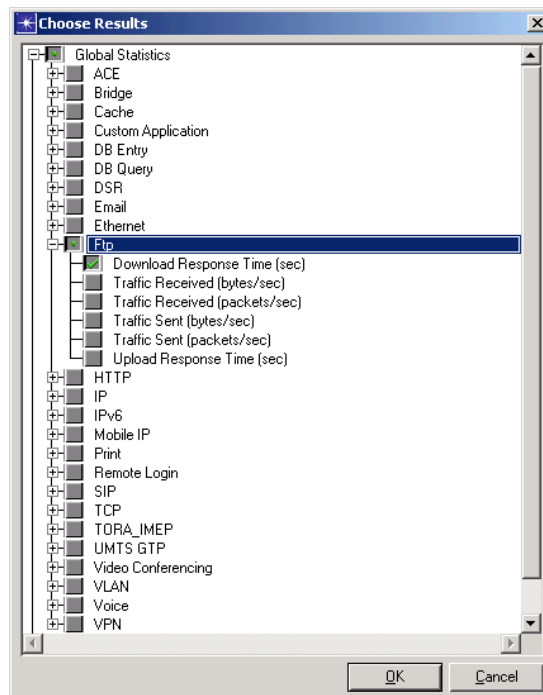
5 Adapt the procedure above to specify the value for the **Application: Supported Services** attribute.

- 5.1 Click in the **Value** cell of the Application: Supported Services row in the Attribute Criteria table and select **Edit...**
 - The **Application: Supported Services Table** dialog box opens.
- 5.2 Add a row to the table.
- 5.3 Click on the **None** text string in the **Name** column, select **Edit...**, and type **File Transfer (Light)**.
- 5.4 Click **OK** to close the **Application: Supported Services** dialog box.
- 6 Click **OK** in the **Logical Object Selection** dialog box.
 - The six LAN objects are selected. You conclude that the FTP (Light) profile is assigned to each client in the LAN, and that the application File Transfer (Light) is supported at the LAN's server.
- 7 Click once in the workspace to deselect the LANs.

Next, review the statistics specified for collection, as follows:

- 1 Right-click in the workspace and select **Choose Individual DES Statistics** from the pop-up menu.
 - ➔ Notice the green dot in the checkboxes next to **Global Statistics** and **Link Statistics**. This indicates that statistics in these hierarchies have been chosen for collection.
- 2 Verify that the following statistics are selected by expanding the hierarchies until you see them, with a green dot in the checkbox.
 - **Global Statistics > Ftp > Download Response Time (sec)**
 - **Link Statistics > point-to-point > utilization -->**

Choose Individual DES Statistics Dialog Box



- 3 Click **OK** to close the **Choose Results** dialog box.

Importing the Traffic Archive

Now that you are familiar with the ready-made model, you can import end-to-end background traffic from a captured traffic archive.

To import a traffic archive, you must do the following:

- 1 Capture or create the traffic (this has already been done for you).
- 2 Configure IT Guru to locate the traffic archive file. This directory is defined by the **traffic_archive_dir** preference.
- 3 Import the traffic into a scenario.

This section covers items 2 and 3.

The exact location of the traffic archive directory differs according to platform. Click to view the procedure that applies to the platform you are using:

- [Windows Configuration](#)
- [UNIX Configuration](#)

Windows Configuration

To configure IT Guru to locate the traffic archive file in Windows:

- 1 Select **Edit > Preferences**.
- 2 In the Find field, type **traffic_archive** and click **Find**.
 - ➔ The **traffic_archive_dir** preference appears.
- 3 Click in the **Value** column to expand it.
Make sure this preference is set to **<reldir>\models\std\tutorial_req\basic\traffic**.
- 4 Click **OK** to close the **Preferences** dialog box.

<reldir> describes the directory that contains the current IT Guru software. You can find your **<reldir>** directory by doing the following:

- 1 Select **Help > About This Application** in the main menu.
 - ➔ The About IT Guru window appears.
- 2 Click on the **Environment** tab.

- 3 Expand the System Information line, then find the **OPNET Release directory** entry. This path is the <reldir> on the host system.

The <reldir> path for a Windows computer is typically

C:\Program Files\OPNET\<release_number>

- 4 Continue with Import Procedure.

UNIX Configuration

To configure IT Guru to locate the traffic archive file in UNIX:

- 1 Select **Edit > Preferences**.
- 2 In the Find field, type **traffic_archive** and click **Find**.
 - ➔ The **traffic_archive_dir** preference appears.
- 3 Click in the **Value** column to expand it.

Make sure this preference is set to
<reldir>/models/std/tutorial_req/basic/traffic.
- 4 Close the **traffic_archive_dir** and **Preferences** dialog boxes.

<reldir> describes the directory that contains the current IT Guru software. You can find your **<reldir>** directory by choosing **Help > About This Application**, then clicking on the Environment tab. Under System Information, find the **OPNET Release directory** entry.

The suggested **<reldir>** for a UNIX system is **/usr/opnet/<release_number>**.

Import Procedure

After the traffic archive files have been placed in the correct directory, they can be imported into IT Guru and applied to the network in the current scenario.

1 Choose Traffic > Import Traffic Flows > From Text (.tr1, .tr2) Files...

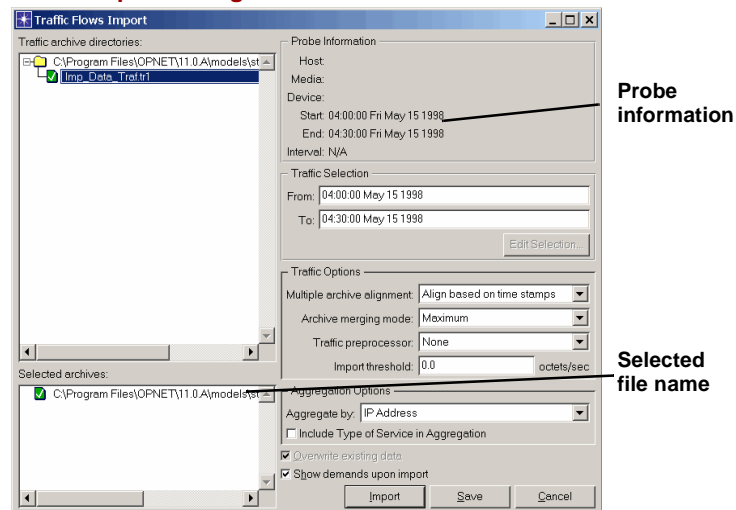
➔ The Traffic Import dialog box appears.

2 A treeview of directories appears in the upper-left pane. Expand the hierarchy to display the directory where your archive files reside (<reldir>\models\std\tutorial_req\basic\traffic)

3 From the list of available files, select Imp_Data_Traf.tr1.

Probe information appears in the Start and End fields and the file name appears in the Selected archives pane.

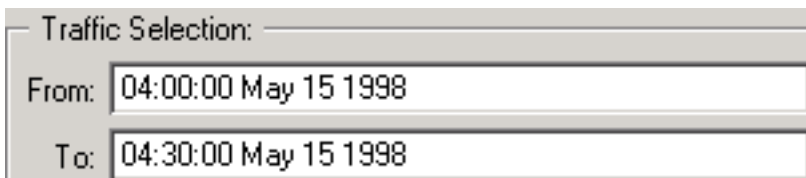
Traffic Import Dialog Box



When importing a traffic file, you can specify several different settings, including the time period of the traffic you want to import.

For this lesson, we will use the entire traffic file, which was collected from **04:00–04:30**. Note that this time period is set in the **From** and **To** options in the Traffic Selection section of the dialog box. You can type different times in the **From** and **To** fields if you need to use a subset of the entire traffic collection period.

Traffic Selection Section of Traffic Import Dialog Box



Traffic Selection:

From: 04:00:00 May 15 1998

To: 04:30:00 May 15 1998

Another attribute that can be set during import is the **Archive merging mode**. This specifies how IT Guru should handle duplicate traffic.

The Archive Merging Mode option specifies how multiple archives of the same traffic data will be handled. Say, for example, that you collected data for the following traffic flows at the same time:

- Archive 1 comes from a probe on node 1 that collects traffic sent to node 2, and
- Archive 2 comes from a probe on node 2 that collects traffic received from node 1

If both probes report the same value, it does not matter if you choose maximum or minimum mode: both modes report the same value. However, if these probes report different values, the specified mode determines which value is used:

- **Maximum** means the larger amount of traffic is used
- **Minimum** means the smaller amount of traffic is used

In some cases, the **Addition** mode is the correct choice. Say, for example, a flow from node 1 to node 2 is split into pieces and sent on multiple paths (one through node A, another through node B).

- If you are collecting traffic only at node 1 or node 2, use either the minimum or maximum mode.
- But if you are, instead, collecting traffic at intermediate nodes A and B, then use the addition mode to correctly consolidate the traffic.

In this case, you will use the most common mode.

- 1 Make sure **Archive merging mode** is set to **Maximum**.

Archive Merging Mode Section of Traffic Import Dialog Box

Multiple archive alignment:	Align based on time stamps
Archive merging mode:	Maximum
Traffic preprocessor:	None

2 Click Import.

- ➔ A window appears and shows statistics about the import.

Traffic Flow Import Statistics Window

Traffic Flow Import Statistics	
Traffic Span Imported	04:00:00.000 May 15 1998 to 04:30:00.000
Network Start Time	04:00:00.000 May 15 1998
Total Number of Files	1
Number of Files with Errors	0
Number of Files Not Imported	0
Total Number of Lines	1,804
Number of Lines with Errors	0
Number of Flows Imported	30
Total Traffic Imported	10,000Gb
Total Packets Imported	10,003,080,000
Files Imported without Errors	C:\PROGRAM~1\OPNET\11.0.A\models\etc
Files Imported with Errors	<Empty>
Files Not Imported	<Empty>

- ➔ A set of blue dashed lines (with arrowheads at both ends) appears in the network. These objects are traffic flows, which model end-to-end background traffic. Each pair of LANs has two connecting traffic flows, each of which specifies traffic from a single source to a single destination.

Sometimes during import, IT Guru cannot associate all the imported traffic with the appropriate network objects. The **Unrecognized Traffic Assistant** can be used to match traffic with the appropriate source or destination nodes (endpoints).

During import, IT Guru matches imported traffic with endpoints in the network using the node's network alias (usually the node's IP address).

Sometimes IT Guru cannot match endpoints to network objects for one of the following reasons:

- 1) the node was omitted from the network
- 2) the traffic endpoint is not actually part of the network

In this import, all traffic is matched to an endpoint without a problem. Had a problem occurred, unmatched traffic endpoints can be associated with a network object using the **Unrecognized Traffic Assistant**. For more information about this feature, see the *Importing Network Traffic* chapter of the *User Guide*.

A traffic flow appears as an arrow that connects two nodes, and specifies end-to-end-traffic between those nodes. You can import traffic flows from external programs and files or you can create them manually.

Each traffic flow contains one or more attributes that describe the traffic in that flow.

The OPNET model library allows you to create “demand” objects in a network. Traffic flow objects are a type of demand object. Demand objects are used to specify demands for network resources such as traffic demands or reserved pipelines.

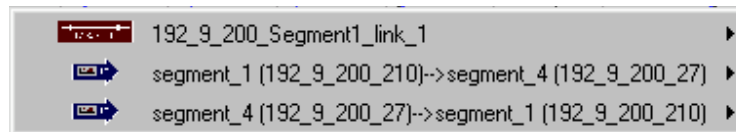
Traffic Flows

In this section we will examine the traffic you just imported. In the process, you will learn how OPNET models end-to-end background traffic using traffic flow objects.

View the Traffic in a Traffic Flow Object

When you imported traffic into the network, OPNET created traffic flow objects (blue dashed arrows) to represent this traffic. In this section, we'll examine one of these objects.

- 1 Right-click on the blue dashed line between the segment_1 LAN (top right) and the center switch.
 - A pop-up menu appears and displays submenus for three different objects: the link object (brown rectangle) and two traffic flow objects (blue arrows).

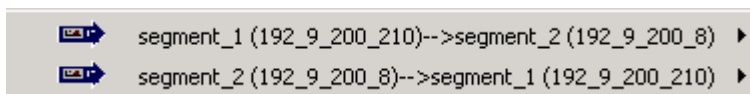


OPNET uses lines to represent relationships between nodes, such as blue dashed lines (traffic flows) for end-to-end traffic and solid black lines (links) for cables and other physical connections. A link or flow object might be hidden by other objects. To view all objects at a specific point in the network, right-click and examine the pop-up menu.

- 2 Click in the workspace to close the pop-up menu.

- 3 Right-click on the blue arrow between the segment_1 LAN and the segment_2 LAN.

➤ A pop-up menu appears and displays submenus for two different traffic flow objects. One specifies traffic from segment_1 to segment_2, and the other specifies traffic from segment_2 to segment_1. Because segment_1 and segment_2 are not connected directly, a link submenu does not appear.



Although what you see looks like a blue dashed line with two arrowheads, it is two overlapping flow objects; each flow has a single arrowhead to indicate the direction of the traffic. (The underlying model for these flows supports unidirectional traffic only.)

- 4 Click in the project workspace.
- 5 Open the Edit Attributes dialog box for the segment_1 --> segment_2 traffic flow:
 - 5.1 Right-click on the traffic flow between the segment_1 and segment_2 LAN.

UNIX: Do not release the right mouse button.

5.2 Select the segment1-->segment_2 traffic flow. This opens the context-sensitive menu for this object.



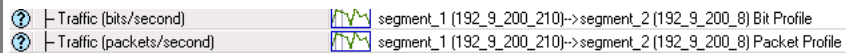
5.3 Select the Edit Attributes menu item. (UNIX: Release the right mouse button.)

➔ The Attributes dialog box for this traffic flow opens.

The model (ip_traffic_flow) used to create this flow object has two attributes that specify the source-to-destination traffic:

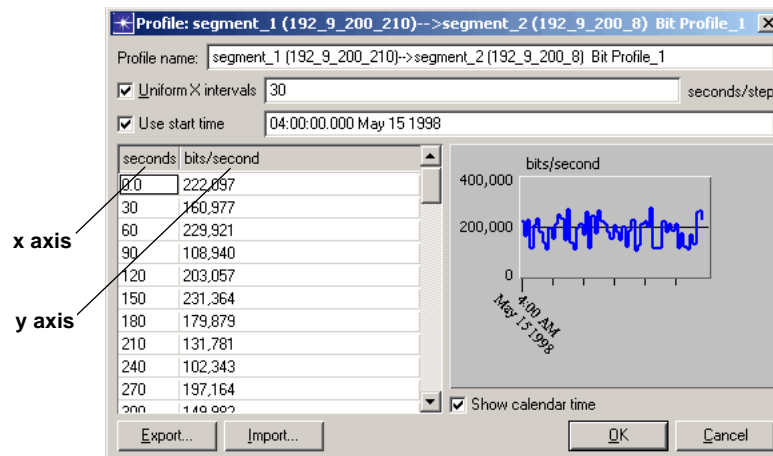
- **Traffic (bits/second)**
- **Traffic (packets/second)**

These attributes describe the traffic data you just imported.



- 6 Click in the **Value** field of the **Traffic (bits/second)** attribute and choose **Edit...** from the pull-down menu.

➔ The **Profile** attribute dialog box opens.



Each flow object contains one or more traffic profiles that specify the rate of traffic for that flow. A "profile" is an attribute that specifies data in an x-y format. In this case, the x scale specifies time intervals in seconds and the y scale specifies the traffic rate in bits/second over each interval.

- 7 Click **OK** to close the **Profile** dialog box, then **OK** to close the Attributes dialog box.

Organizational Scaling

Now that you have a network loaded with traffic, you can begin to explore what will happen as the organization grows and traffic increases, first by 50 percent and then by 75 percent.

IT Guru can be used to model organizational scaling quickly and easily.

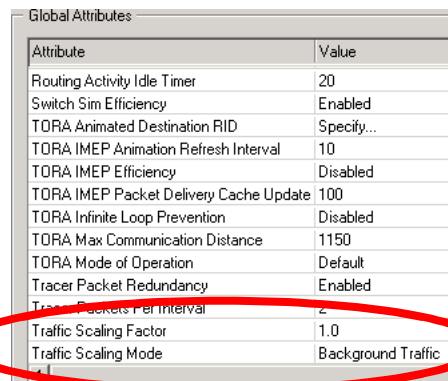
First, you need a baseline.

Run the Simulation

- 1 Choose **DES > Configure/Run Discrete Event Simulation**.
- 2 Click on the **Common** item in the treeview and verify that the **Duration** is 360 seconds.
- 3 Click on **Inputs > Global Attributes** and verify that
 - The **Traffic Scaling Factor** is 1.0
 - The **Traffic Scaling Mode** is Background Traffic

(You might need to scroll down in the Global Attributes table to see these attributes.)

Configure Discrete Event Simulation Dialog Box



Attribute	Value
Routing Activity Idle Timer	20
Switch Sim Efficiency	Enabled
TORA Animated Destination RID	Specify...
TORA IMEP Animation Refresh Interval	10
TORA IMEP Efficiency	Disabled
TORA IMEP Packet Delivery Cache Update	100
TORA Infinite Loop Prevention	Disabled
TORA Max Communication Distance	1150
TORA Mode of Operation	Default
Tracer Packet Redundancy	Enabled
Tracer Packets Per Interval	2
Traffic Scaling Factor	1.0
Traffic Scaling Mode	Background Traffic

4 Click Run.

- The simulation begins execution. When it is finished, click the **Close** button in the Simulation Sequence dialog box.
- If you had problems, see "*Troubleshooting IT Guru Tutorials*".

You have generated a baseline for the results, but don't look at them just yet. Our interest is in seeing the effect of traffic growth, so we will look at these statistics when we can compare the results among all three scenarios.

Your task now is to create additional scenarios in which the background traffic is increased by 50 percent and 75 percent. The best way to do this is to duplicate the original scenario and then scale the traffic for each additional scenario.

For Traffic Increased by 50 Percent

- 1 Choose **Scenarios > Duplicate Scenario...**
- 2 Name the new scenario **scaled_50_percent**.
- 3 In the new scenario, choose **DES > Configure/Run Discrete Event Simulation**.
 - 3.1 Click **Inputs > Global Attributes** in the tree view and change the **Traffic Scaling Factor** simulation attribute to **1.5**.
 - 3.2 Verify that the **Traffic Scaling Mode** is set to Background Traffic (that is, flow traffic and baseline load traffic, but not explicitly generated traffic).

Traffic Scaling Factor is Set to 1.5

Traffic Scaling Factor	1.5
Traffic Scaling Mode	Background Traffic
- 4 Click **Apply** to save your DES configuration and then click **Cancel**.
- 5 Save the project by choosing **File > Save**.

For Traffic Increased by 75 Percent

Follow the same method:

- 1 Choose **Scenarios > Switch To Scenario > original_traffic**.
- 2 Duplicate the scenario and name the new scenario **scaled_75_percent**.
- 3 In the new scenario, choose **DES > Configure/Run Discrete Event Simulation**.
 - 3.1 Change the **Traffic Scaling Factor** simulation attribute to **1.75**.
 - 3.2 Verify that the **Traffic Scaling Mode** is set to Background Traffic.

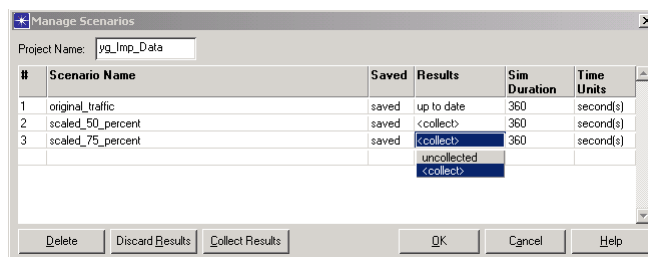
The next time you run a discrete event simulation, the background traffic in the network will be increased by 75 percent.
- 4 Click **Apply** to save your DES configuration and then click **Cancel**.
- 5 Save the project by choosing **File > Save**.

You can run simulations for multiple scenarios with a single command from the **Manage Scenarios...** dialog box. You can also duplicate, delete, and rename scenarios.

Now that each scenario has been created with the appropriately scaled traffic, you can run the two new scenarios as a batch.

- 1 Choose **Scenarios > Manage Scenarios...**
 - ➔ Notice that the **Results** for the **original_traffic** scenario are shown as **up to date**, while they are **uncollected** for the two new scenarios.
- 2 In the **Results** field, change **scaled_50_percent** and **scaled_75_percent** to **<collect>**.

Running Simulations from the Manage Scenarios... Dialog Box



3 Click OK.

- The simulations begin execution. The two simulations should take about one minute to run. Close the Simulation Sequence dialog box when complete.
- If you had problems, see "*Troubleshooting IT Guru Tutorials*".

Now that all the results have been collected, you can view utilization on the links and FTP download response time to see how the network performs under various loads. To view utilization:

- 1** Right-click in the workspace and choose **Compare Results** from the pop-up menu.
- 2** Verify that the **Discrete Event Graphs** tab is selected.
- 3** Expand **Object Statistics > 192_9_200 > 192_9_200_Segment1_link_0[0] > point-to-point**. Select **utilization --->**.

The utilization statistic will be more meaningful if viewed as the average utilization over time. You can apply a filter to the results to view the average utilization:

- 1 At the bottom of the Compare Results dialog box, click on the filter pull-down menu (it shows the default value, **As Is**) and select **average**.

Compare Results Dialog Box (detail)

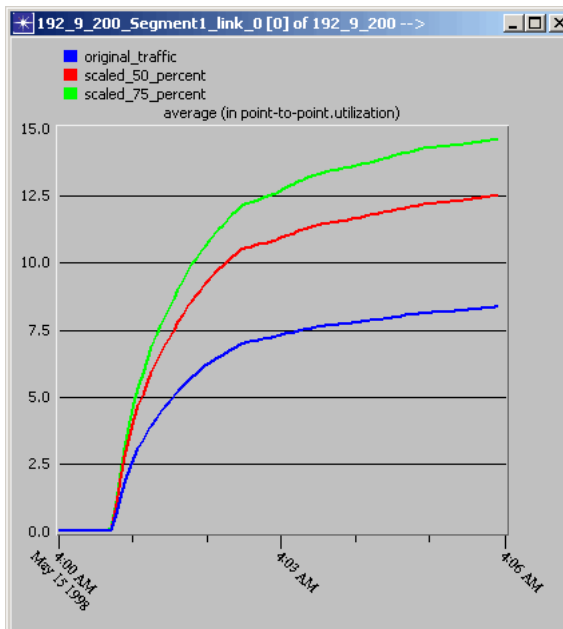


Set the filter pull-down menu to average

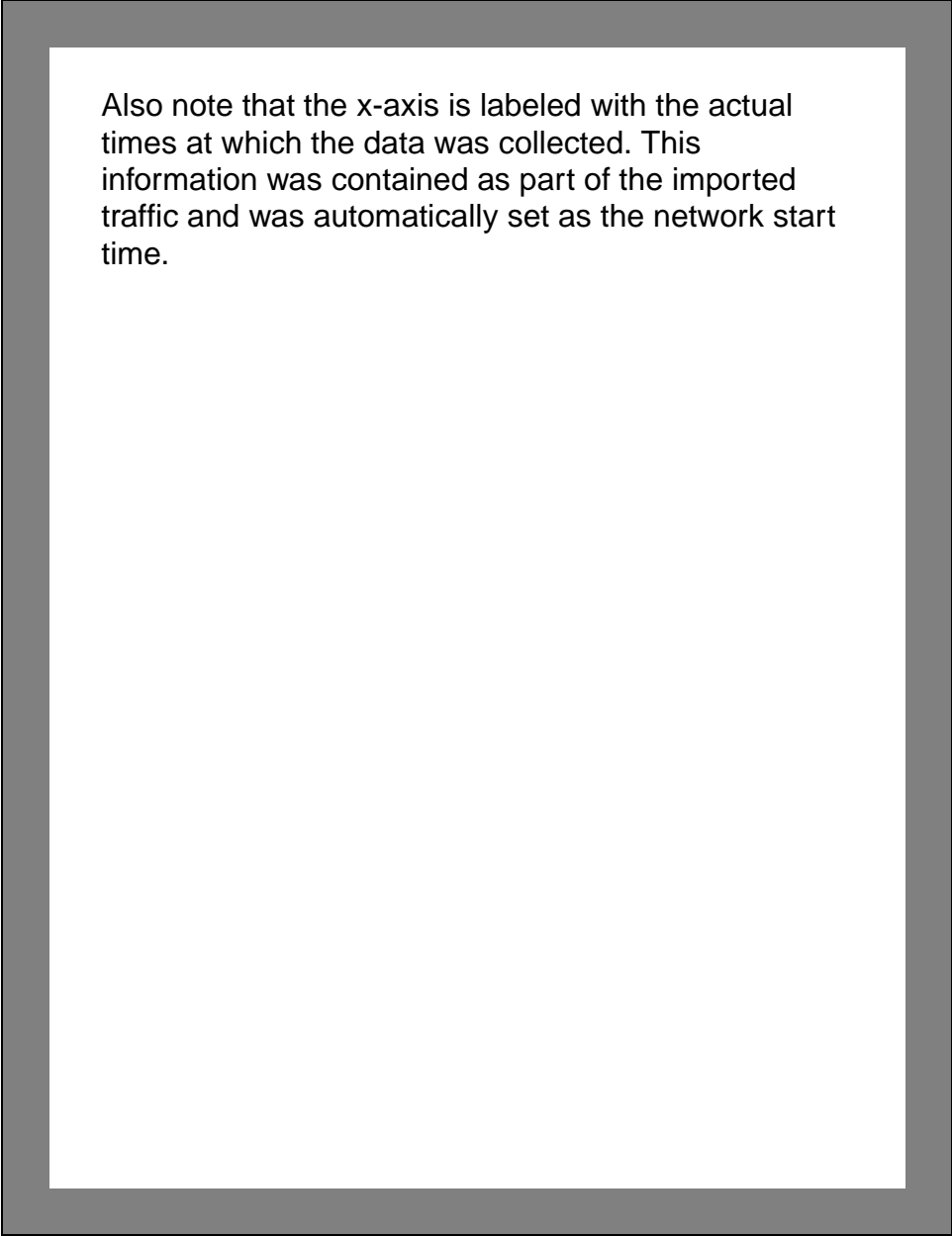
- 2 Click **Show** to display the graph.
 - The utilization in each scenario for the 192_9_200_Segment1_link_0 object is graphed.

The graph for average utilization should be similar to the following figure:

Average Utilization Graph



Note that although utilization has not yet reached steady state, its low value (less than 15 percent for the **scaled_75_percent** scenario) indicates that the between-LAN links are not the bottleneck in this network.

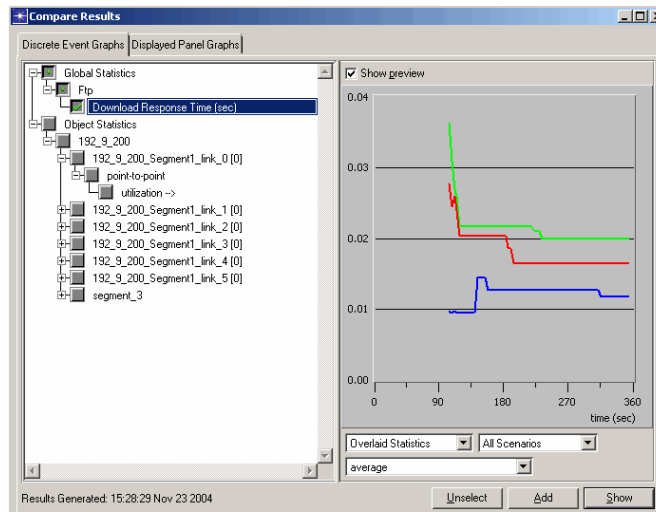


Also note that the x-axis is labeled with the actual times at which the data was collected. This information was contained as part of the imported traffic and was automatically set as the network start time.

The other statistic you collected was Download Response Time:

- 1 Move the Utilization graph out of the way, but don't close it.
- 2 In the Compare Results dialog box, click the **Unselect** button.
- 3 Expand **Global Statistics > Ftp**. Select **Download Response Time (sec)**.

Compare Results Dialog Box, Filter Set to Average

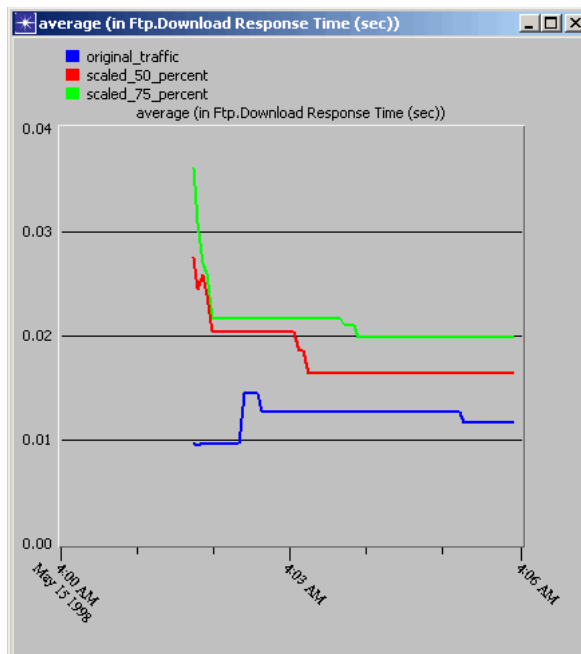


- 4 Verify that the graph filter is set to **average**.

5 Click on the **Show** button.

➔ The resulting graph should resemble the following figure:

Average of FTP Download Response Time Graph



6 Notice the following:

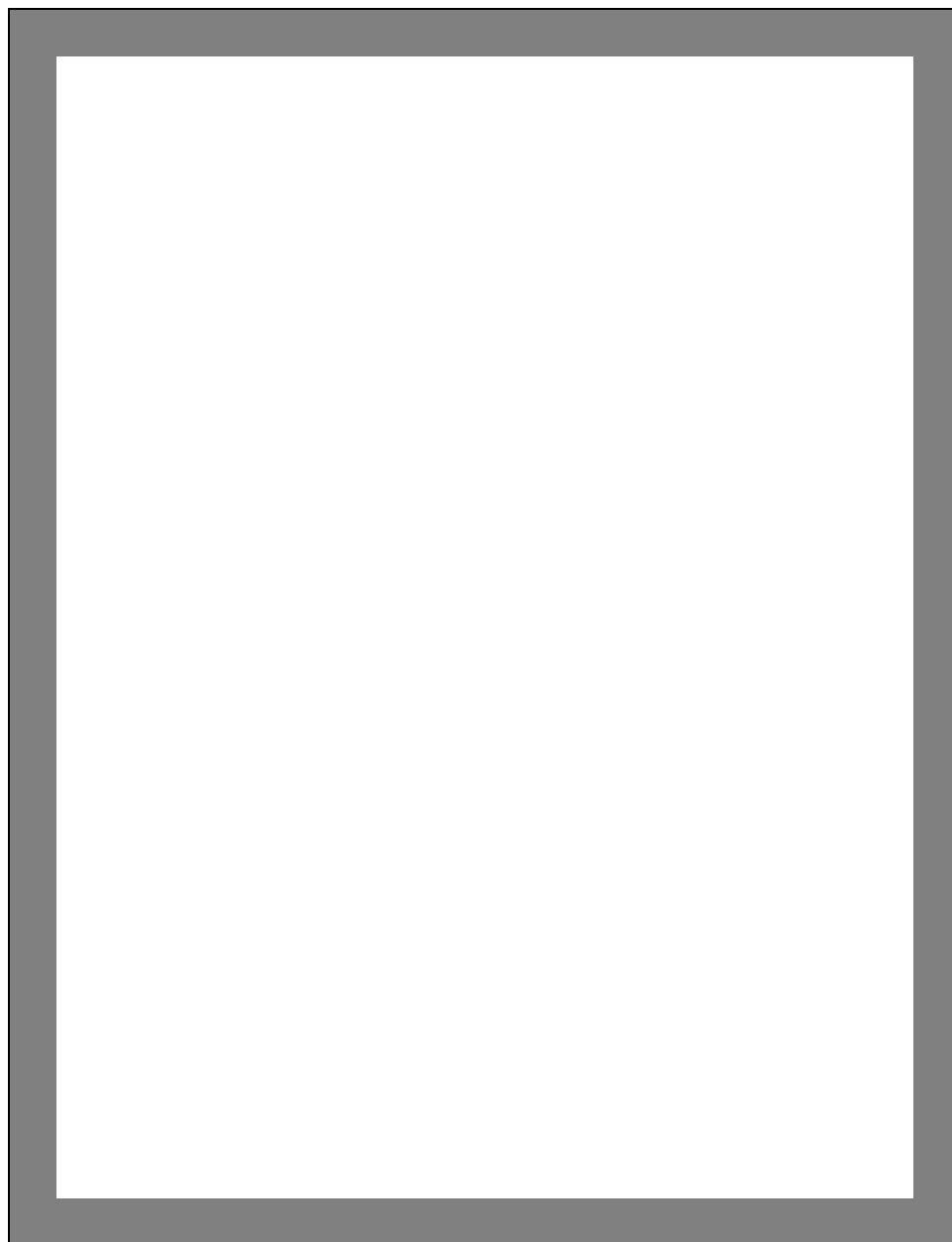
- Download response time increases when traffic is scaled 50 percent.

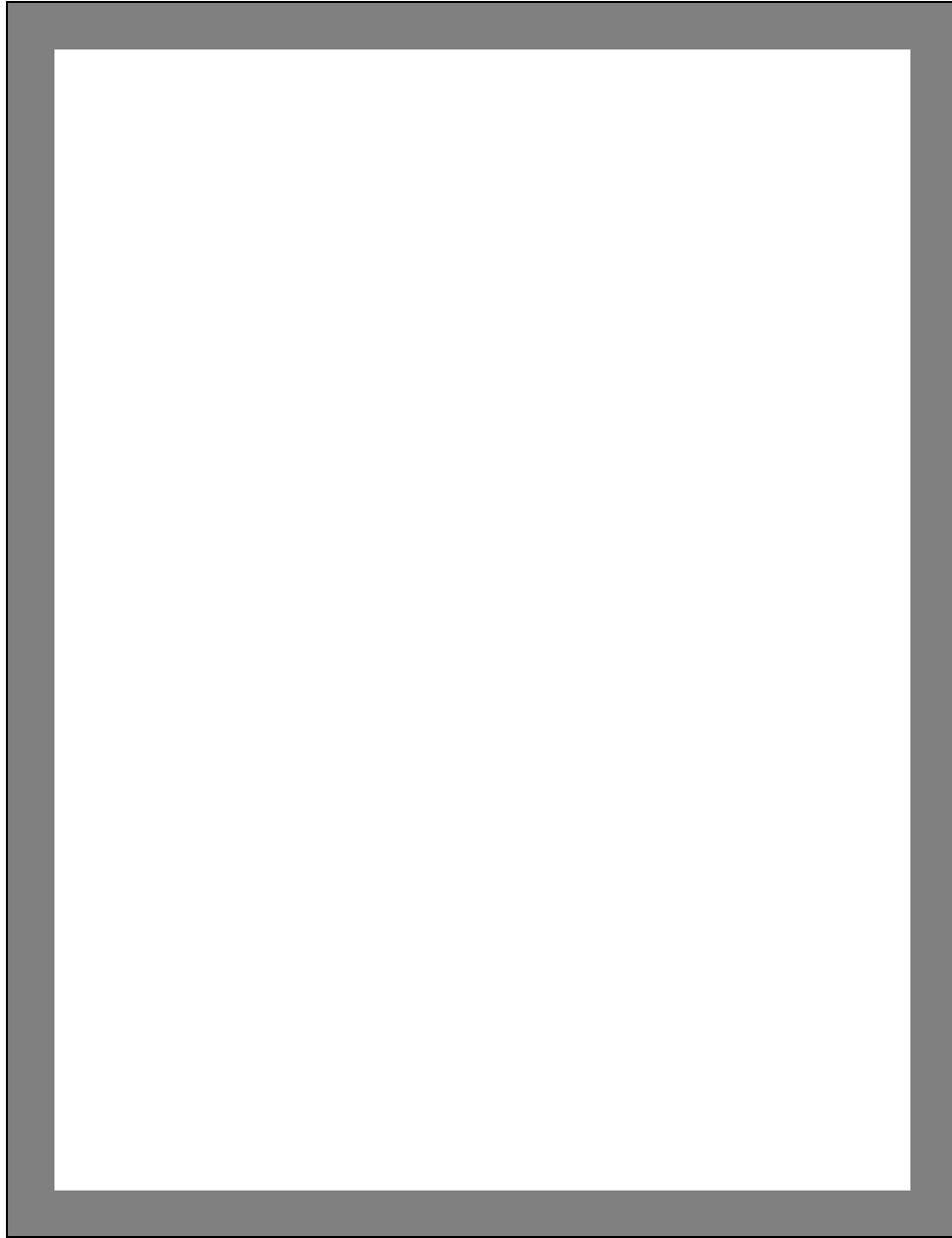
- Response time continues to increase when traffic is scaled up to 75 percent.
- But the increases are very small: far less than one second.

Although these increases are minor, you have run only one simulation set. For the results to be statistically useful, you might want to try different types of explicit traffic and vary the simulation seed.

You have now completed the Importing Traffic lesson.

- Select **File > Save**. We suggest saving the project so you can refer to it again if needed.
- Return to the Tutorials main menu and continue with the Expert Service Prediction (ESP) tutorial.





6 Expert Service Prediction

Expert Service Prediction

Overview

This lesson explores practical applications of IT Guru's Expert Service Prediction (ESP) features for service level management, including:

- Specifying **service level agreement (SLA)** thresholds for different scenarios
- Configuring simulations to study the impact of network traffic growth
- Generating and interpreting Web reports

An SLA typically specifies certain levels of service—server response time, router-processing delay, link utilization—that the network provider promises. Within IT Guru, an SLA is a set of thresholds that IT Guru applies to statistics you specify. IT Guru then shows violations of those thresholds in graphs and Web reports.

In the scenario used for this tutorial, TENPO software company has six LANs in their main office, all connected to a single switch. Their Information Systems (IS) group must guarantee a response time of less than 11 seconds among the LANs during

system backups when the network load is greatest. The IS group has already imported a LAN-level model of their network and imported baseline background traffic.

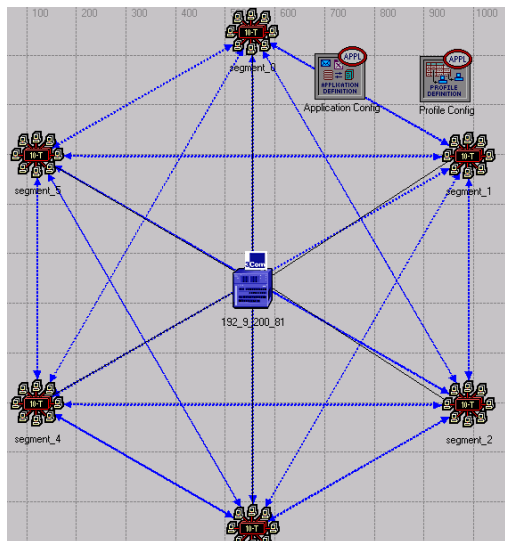
Your job is to take the existing models and specify SLA thresholds, and then run simulations to determine how long it will be before the network, at its current growth level, will violate its SLA. Your supervisor wants a report of your findings.

Examining Traffic in the Existing Model

To load the existing model that the IS group has already created, follow these steps:

- 1 If IT Guru is not already running, start it.
- 2 Select **File > Open**, and then choose `<reldir>\models\std\tutorial_req\basic`.
- 3 Choose **ESP_tutorial** from the Files: list, and then click **Open**.
➔ The network model opens in the Project Editor.

The Existing Model



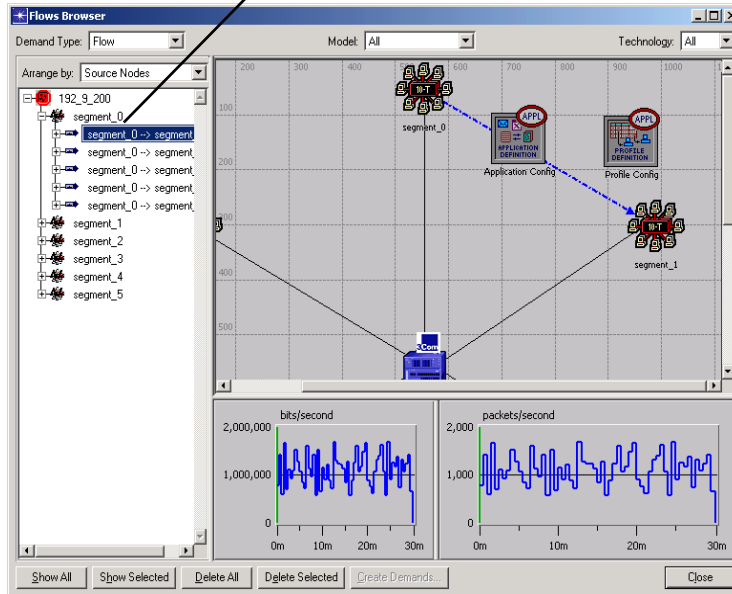
- 4 Select **File > Save As...**, and then save the network as **<your_initials>_ESP_tutorial** in your **<home>\op_models** directory.

You can now examine the existing background traffic on the network.

- 1 Select **Traffic > Open Flows Browser**.
- 2 Examine the traffic associated with **segment_0**.
 - 2.1 Expand **segment_0**.
 - 2.2 Select the flow **segment_0 --> segment_1**.
 - Two graphs display under the network model. The graphs show the traffic for the selected flow, which now appears blue in the model.

Flows Browser Dialog Box

Expand segment_0 in the Source Nodes pane, then select segment_0->segment_1



The graph on the left shows traffic in bits per second during a 30-minute period. It shows traffic ranging from about 600,000 to 1,700,000 bits per second.

The graph on the right shows traffic in packets per second during the same period. It shows traffic ranging from about 600 to 1,700 packets per second.

- 3 You can repeat the procedure to examine traffic for other segments.
- 4 Close the Flows Browser.

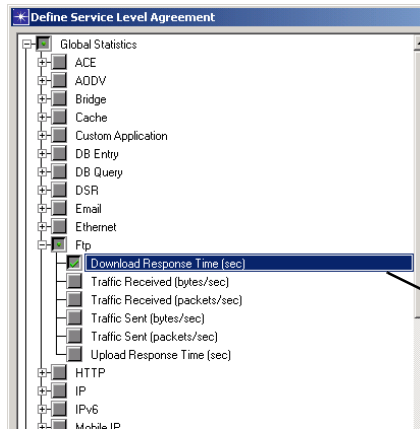
Setting SLA Thresholds

Now that you have reviewed the IS network model, your goal is to set an SLA that will tell you if the IS guarantee is being violated. For TENPO's network, IS has guaranteed an FTP response time of 11 seconds.

To set the SLA thresholds, follow these steps:

- 1 Select **DES > Expert Service Prediction > Define Service Level Agreement...**
- 2 From the list of available statistics, choose **Global Statistics > Ftp > Download Response Time (sec)**.

Define Service Level Agreement Dialog Box



Choose Global
Statistics: Ftp:
Download Response
Time (sec)

3 Define the SLA as shown in the following figure:

Defining the Download Response Time SLA

Show compliance if the "Download Response Time (sec)"

statistic is

percent of the time.

Bucket duration:

"11" means "11 seconds."
The unit of measure is
taken from the statistic (in
this case, response time).

➔ FTP response time is in compliance if more than 50 percent of the response times, in any 1-minute period, are 11 seconds or less. So IT Guru reports an SLA violation if more than 50 percent of the response times, in any 1-minute period, are above 11 seconds.

4 Save the SLA for Download Response Time, naming it **<initials>_11_sec_resp_time**.

Next, specify an SLA for utilization:

- 1 Click the **Clear** button.
- 2 Select the following statistic:
Link Statistics > point-to-point > utilization →
- 3 Define the SLA as shown in the following figure:

Defining the Utilization SLA

Show compliance if the "utilization -->"

statistic is: below 75

50 percent of the time.

Bucket duration: 1.0 Minute(s)

“75” means 75 percent.
The unit of measure is taken from the statistic (in this case, utilization rate).

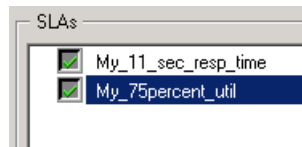
Utilization is in compliance if utilization for a link is lower than 75 percent for more than half of a 1-minute period. Therefore, IT Guru will report an SLA violation if utilization for a link is higher than 75 percent for more than half of a 1-minute period.

- 4 Click **Save** when you are done. Name the SLA **<initials>_75percent_util**, and then close the Define Service Level Agreement dialog box.

Now that you have defined the response time and utilization SLAs, you must apply them to the scenario.

- 1 Select **DES > Configure/Run Discrete Event Simulation....**
- 2 Expand the **Outputs** treeview branch and click on **Outputs > Reports**.
- 3 From the list of available SLAs, check **<initials>_11_sec_resp_time** and **<initials>_75percent_util**.

Applying the SLA



Note: You can also use the **Define SLA Report...** button to define reports.

For your supervisor, you will generate a Web report. To specify the results contained in the report, perform the following steps:

- 1 In the **Statistics Reports** section, click the **Define Statistics Report...** button.
- 2 Enable the **New Report** radio button, and then click **OK**.
- 3 From the available statistics, select the following:
 - **Global Statistics > Ftp > Download Response Time (sec)**
 - **Link Statistics > point-to-point > utilization —>**
 - **Link Statistics > point-to-point > utilization <—**
- 4 When you have finished selecting statistics for the Web report, click **Save**, and then name the report **<initials>_ESP_report**.

Note that you can reuse any report template, including predefined templates, in other scenarios.

Now that you have defined the report template, you must apply it to the scenario:

- 1 In the **Statistics Reports** section, from the list of available reports, choose **<initials>_ESP_report**.
- 2 Click on the **Common** treeview branch.
- 3 Verify that the **Duration** field is set to **15 minutes**.
- 4 Expand the **Inputs** treeview and click on **Inputs > Global Attributes**.
- 5 Verify that the simulation attribute **Traffic Scaling Mode** is set to **All Traffic**.

Do not close the Configure/Run DES dialog box.

Specifying Traffic Growth

Instead of creating multiple scenarios and scaling the traffic by hand, you can use the **Specify Traffic Growth** feature to scale traffic growth automatically within a single scenario.

Now that you have created a baseline scenario and defined the SLAs for response time and utilization, you can use the Specify Traffic Growth feature to study the effects of increased traffic on the network. You can define either simple or compound traffic growth without having to duplicate the baseline scenario or scale the network traffic manually.

To scale the traffic growth for this scenario, follow these steps:

- 1 In the treeview of the Configure/Run DES dialog box, select **Inputs > Traffic Growth**.
- 2 In the **Traffic Growth** page, select **4** in the **Iterations** field and select **2** in the **Months** field. You will study TENPO's traffic growth over the next 6 months in 2-month intervals.

Traffic Growth Page

Number of runs in set: 4 Save vector and environment files for each run in set
 Pause between each run in set

Traffic Growth

Run: 4 iterations, each separated by: 2 months

Use growth rate of: 15 % per iteration
 Use compound growth

Start Time	Growth Rate (%)	Traffic Level (%)
Now		100.00
Now + 2 months	15.00	115.00
Now + 4 months	15.00	132.25
Now + 6 months	15.00	152.09

Store only summary results

Run Cancel Apply Help

- 3 Enable the **Use growth rate of** checkbox and set the value to **15%** per iteration.
- 4 Enable the **Use compound growth** checkbox.
For this study, you are only interested in the summary results, so make sure the **Store only summary results** checkbox is selected.
- 5 Click **Run** to begin the simulations.
- 6 Click **Close** when the simulations finish.

Viewing a Web Report

Although you can always view the results of your simulations in IT Guru, the Web report feature allows you to organize and distribute the results of your simulations.

When your simulations have executed, you can view the Web report that includes all results.

To generate the Web report, perform the following steps:

- 1 Select **DES > Results > Generate Web Report....**
 - ➔ The Specify Report dialog box displays.
- 2 Click **OK** to generate a report with the default name.
 - ➔ A Web report is generated and placed in the directory specified by the preference **web_rep_storage_dir**.
If **web_rep_storage_dir** is not set, the Web report is placed in **<user_home_dir>\op_admin\tmp**.

Follow these steps to view the Web report:

1 Choose DES > Results > Launch Last Web Report.

➔ The Select Report dialog box opens, showing a list of reports generated (by name or by date).

(If there is only one Web report, it will open automatically and you can skip the next step.)

2 Left-click on the View Most Recent button.

➔ IT Guru launches the Web browser specified by the **browser_prog** preference and opens the Report Index page. If the browser does not open, check that the **browser_prog** preference specifies the correct path.

There are two types of graphs available for displaying SLA compliance information, depending on whether you ran a single simulation scenario or a multiple simulation scenario.

When the Web report opens, you will see the Index page, which displays several links. These links are described in detail on the following pages.

Web Report Index Page

The screenshot shows a web interface for simulation reports. On the left, a sidebar contains a 'Results' section with three links: 'SLA: My_11_sec_resp_time', 'SLA: My_75percent_util', and 'Report: My_ESP_report'. Below this is a 'Simulation Reports Home' link and a section for 'SLA: My_11_sec_resp_time' with sub-links for 'Violations' (Text Summary, Global Statistics, Ftp) and 'Close Calls' (Text Summary). A 'Link display area' label points to the 'Text Summary' link under 'Close Calls'. On the right, a 'Data display area' label points to a grid of charts and graphs. An inset image shows a person standing in front of a large wall of data displays. Below the charts, text reads 'Project: My_ESP_tutorial Scenario: original_traffic' and 'Simulated from January 2000 to August 2000.' The OPNET logo is at the bottom right.

First, look at the response time global statistic you collected:

- 1 In the Results area, click on the **SLA: <initials>_11_sec_resp_time** link.
 - ➔ Links to information about SLA **violations** and **close calls** display in the lower section of the link display area.

Links on 11_sec_resp_time SLA

SLA: My_11_sec_resp_time

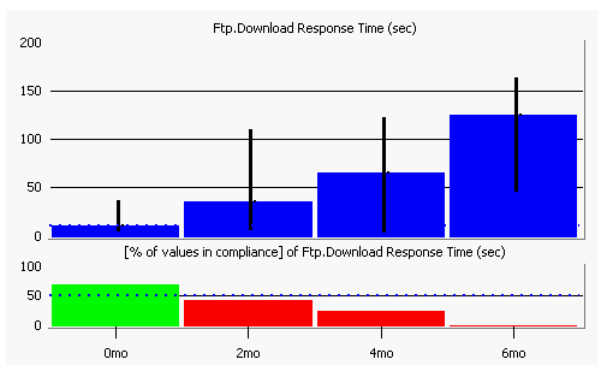
- Violations
 - [Text Summary](#)
 - [Global Statistics](#)
 - [Ftp](#)
- Close Calls
 - [Text Summary](#)

2 Click on the **Violations:Text Summary** link.

- ➔ The text summary indicates that during the worst period of violation, the FTP download response time was above the 11-second SLA nearly all the time. You can view the graph of this SLA by clicking on the **Global Ftp Download Response Time (sec)** link.

The graph of FTP Download Response Time is shown in the following figure. The months shown in the time axis may be different in your graphs because the starting month depends on when you run the simulation:

FTP Download Response Time



Note that the 11-second SLA is violated during 3 simulation periods, beginning 2 months in the future.

Next, look at the link utilization object statistic:

- 1 Click on the **SLA: <initials>_75percent_util** link.
 - ➔ Links to information about **SLA violations** and **close calls**, plus reports on the **top nodes and links** display in the lower section of the link display area.

Links on 75percent_util SLA

SLA: My_75percent_util

- Violations
 - [Text Summary](#)
 - [By Link](#)
 - By Statistic
 - [point-to-point](#)
- Close Calls
 - [Text Summary](#)
- Top Nodes and Links
 - [By Link](#)
 - By Statistic
 - [point-to-point](#)

For information about violations, you can examine the text summary, the summary by link, and the summary by statistic.

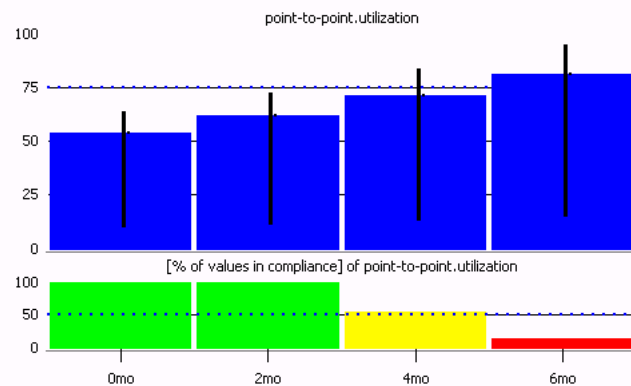
- 1 Click on the **Violations: Text Summary** link.
 - The text summary lists each network link that violated the SLA. Each network link was above the SLA limit during at least one period; some violated the SLA during two periods, and the network link to **segment_0** was above the SLA nearly all the time during its worst period.

2 Click on the link for the **192_9_200_81 <--> segment_2** network link.

➔ The report shows graphs that correspond to the link display.

This network link violated the SLA during two periods. During the worst period, it was above the limit most of the time. Utilization rose steadily over the eight months covered, exceeding the SLA during the third period.

Violations for the Utilization Statistic

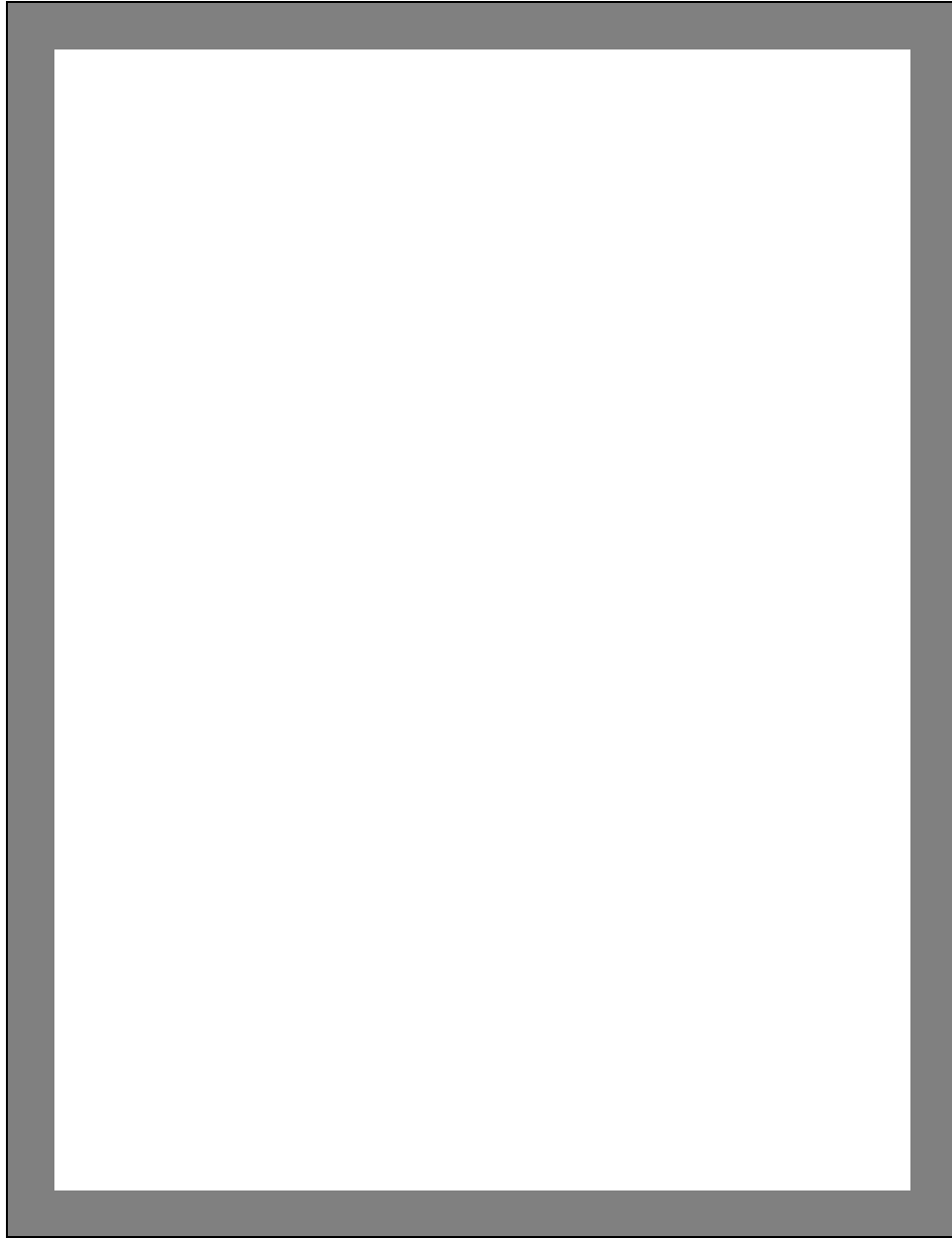


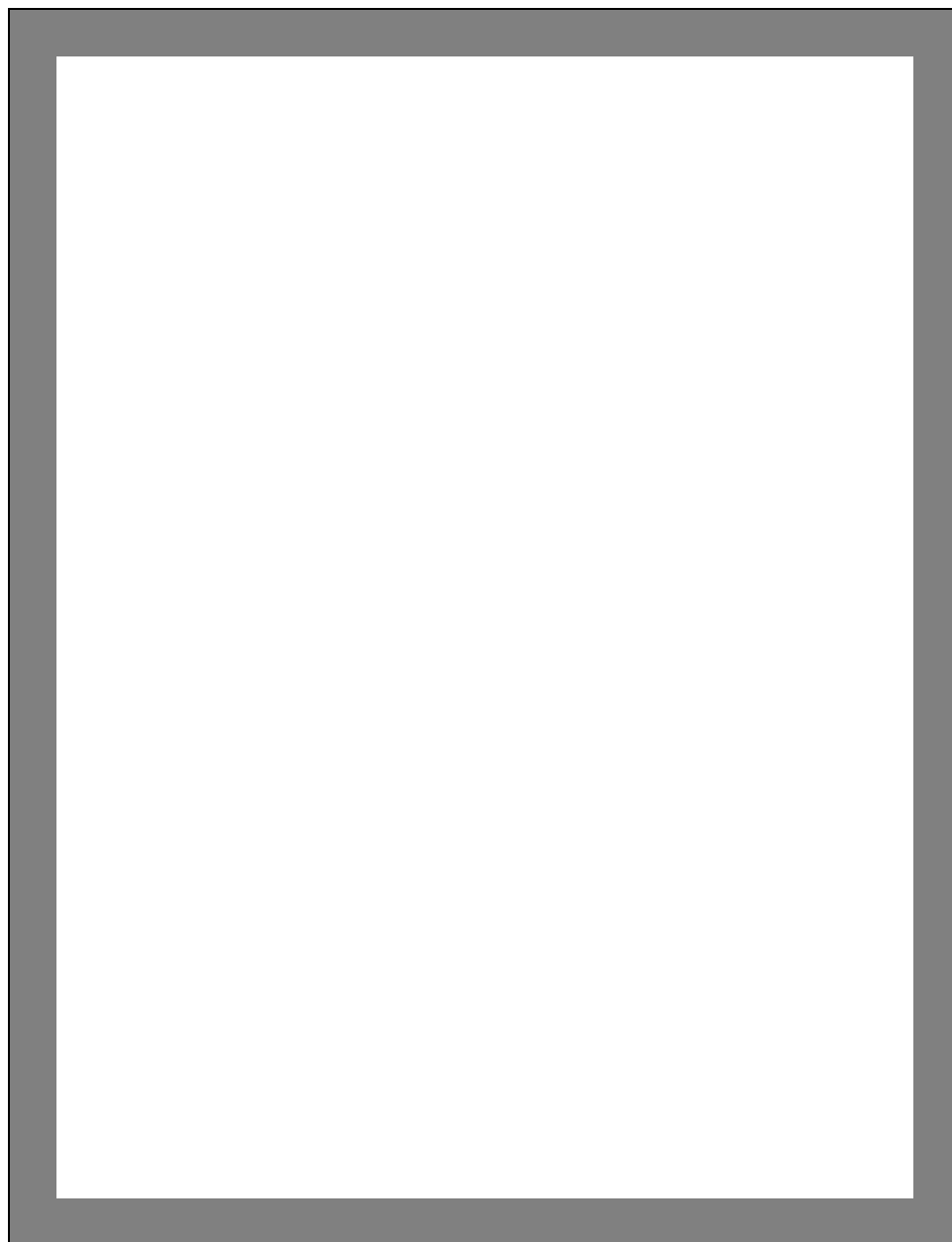
From the data seen so far, you can answer the question initially posed: **If traffic continues to grow at the current rate, how long will it be before an SLA is violated?**

Answer: Both SLAs will be violated two periods from now, in 4 months.

To give your supervisor the comprehensive report requested, you can simply provide the URL to the top-level page in the report.

Return to the main tutorial menu and choose the next tutorial, **Productivity Features**, from the list of available lessons.





7 Productivity Features

Productivity Features

Overview

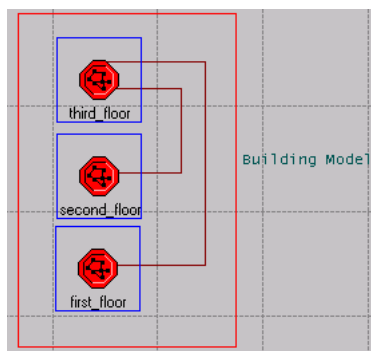
This lesson explores IT Guru productivity features—tools that improve model-building efficiency and usability. This lesson focuses on the following tools:

- Preferences
- Device Creator
- Derived Models
- Logical Object Selection
- Network Browser
- Annotation Palette

This lesson focuses on several special tools, called “Productivity Features,” that you can use to improve your model-building efficiency.

IT Guru has a number of built-in features that have not been used in previous lessons. Although these tools are not essential to model building, they can greatly improve your understanding of models and your modeling efficiency. To familiarize yourself with these tools, you will use each of them to create a network model.

Prod_Features Network Model



Getting Started

For this lesson, you will start with an existing network and use several features to add to and enhance the network.

- 1 Choose **File > Open...** and select **Project** from the **Files of type** pull-down menu.
- 2 Open the **Prod_Features** project, which is located in the following directory:
`<opnet_dir>\models\std\tutorial_req\basic`
➔ The **Prod_Features** project and scenario open in the Project Editor.
- 3 Choose **File > Save As...** and save your project as **<initials>_Prod_Features** in your default model directory.

The **Prod_Features** project contains a network of three subnets, each representing a floor of a building. The first and second floors contain star-topology 10BaseT Ethernet networks. The third floor, which you will build, is the location of the network's two e-mail servers and a switch that connects the three floors to the outside world. You will use Device Creator to create a special switch and two e-mail servers for the third floor.

In addition, a brief overview of customizing the user interface in IT Guru is provided.

To start, you will change the name of the object palette's model list to reflect your initials.

- 1 Open the object palette.
 - The object palette opens with the **Prod_Features_Model_List** model list active.
- 2 Click the **Configure Palette...** button in the object palette.
- 3 Click **Save As...** in the Configure Palette dialog box.
- 4 Save the model list as **<initials>_Prod_Features** in your default model directory.
- 5 Click **OK** to close the Configure Palette dialog.
- 6 Close the object palette.

You can now add or delete models from the model list you created without changing the original model list.

Preferences

Preferences are used to customize IT Guru to your specifications.

You can change preferences to customize many aspects of IT Guru, including the user interface. For example, you can use the following procedure to change the background color of the workspace:

- 1 Choose **Edit > Preferences**.
- 2 Find the **ui_colors.tool_back** preference.
To find this preference, either scroll to its location in the list or type "tool_back" in the edit field at the top, then click **Find** to view a subset of preferences that mention "tool_back."
- 3 Change the **ui_colors.tool_back** preference by clicking in its value field and choosing a background color from the color chooser. Click **OK** when you are done.
- 4 Click **OK** to close the **Preferences**.
 - ➔ The background color has been changed and will take effect after the screen is refreshed (for example, when you open a subnetwork).

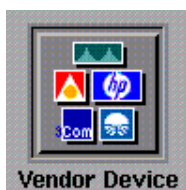
Device Creator

Device Creator lets you customize devices and save them.

We will use this feature to create a new 3Com switch for the third floor of your network. You need to modify the standard 3Com switch to a different port configuration.

- 1 Choose **Topology > Device Creator...**
- 2 In the Device Creator dialog box, select **Vendor Device**.

Vendor Device Icon



➔ The Vendor Device dialog box appears.

- 3 Select **3Com** from the available vendors.
- 4 Select **CoreBuilder 3500** from the available devices.

- 5 In the **Model name** field, change the value to **<initials>_Prod_Features_Switch**.

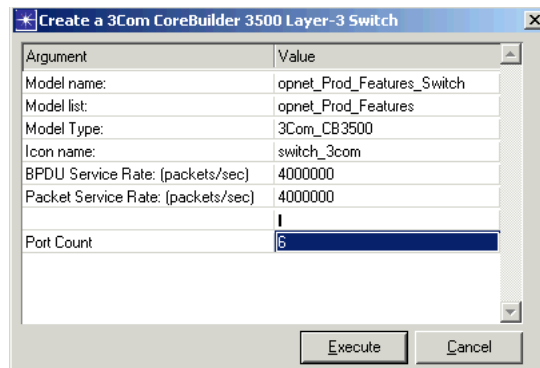
You can also specify a palette for the new device:

In the **Model list** field, change the value to **<initials>_Prod_Features**.

Because you need five 10BaseT ports for the network—one port per server (2), one port for each of the other floors (2), and one port to connect to the outside world (1)—you would probably purchase a 3C35210 card, which has six ports, for the switch. You can specify this in the **Port Count** setting:

- 6 Change the value of **Port Count** to **6**.

Create <device> Dialog Box



7 Click **Execute**.

➤ A message appears when your device is ready.

8 Click **OK** to close the notification dialog box.

Derived Models

You can derive customized network models—called **derived models**—from existing models.

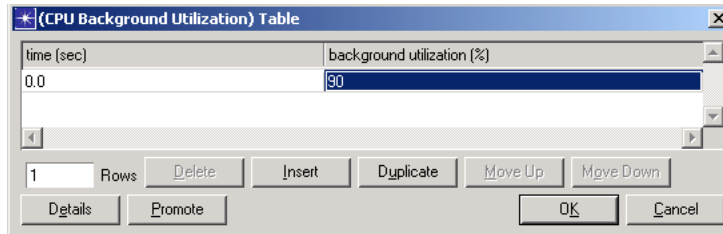
Another way to create a customized model is to derive it from an existing model. This is useful when you have an object with a complex configuration and do not want to configure the object each time you use it. You can create one instance of the object with the necessary configuration and use it wherever necessary. You can also derive models for vendor models that do not come standard with IT Guru.

Standard models often have a suffix modifier that indicates the level of derivation of the model. An advanced model (`_adv` suffix) shows all attributes and sets them to their default values. An intermediate model (`_int` suffix) shows the most important attributes with their most useful values and hides attributes that are unlikely to be changed.

In this section, you will derive a new server from an existing model. The derived server will have over 90 percent of its CPU resources devoted to background processing.

- 1 Open the object palette if it is not already open.
 - The icon of the 3Com CoreBuilder switch that you just created is now in the object palette.
- 2 Right-click on the **ethernet_server** object in the object palette.
 - Right-clicking on a node in the object palette opens its node model description, which contains an object description and other important information. You cannot edit the information in this dialog box.
- 3 Click on the **Derive New** button.
 - The Derive New Model dialog box opens; you can edit the fields in this dialog box.
- 4 In the **CPU Background Utilization** row of the **Attributes** table, select **Edit...** from the list in the **Initial Value** column.
- 5 In the CPU Background Utilization table, change the **background utilization (%)** value from **0.0** to **90**.

Changing the Background Utilization Attribute

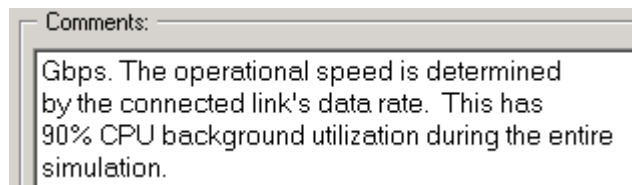


- 6 Click **OK** when finished.

When you create a derived model from another model, you might want to change other aspects of the model, such as the keywords, the icon used to represent the object, and the object description. In this case, you will change the object description only.

- 1 Add a sentence to the end of the first paragraph, noting that this model has 90 percent of its CPU resources in use for the entire simulation.

Sentence Added to Comments



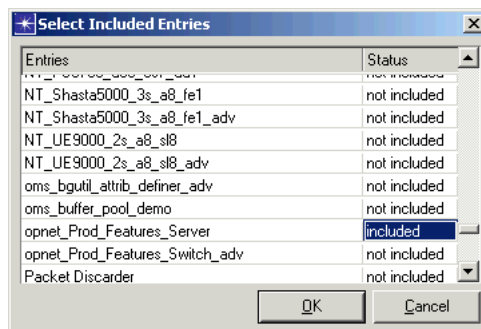
- 2 Click on the **Save...** button when you finish editing the comments.
 - IT Guru prompts you for the filename and target directory of the model.
- 3 Select your default model directory in the **Model directories** treeview, type **<initials>_Prod_Features_Server** in the **File name** field, and click **Save**.
 - Your derived model is saved as **<initials>_Prod_Features_Server**.
- 4 Close the Edit Derived Model dialog box.

Now that you have created the new derived model, you must add it to the object palette.

- 1 Open the object palette if it is not already open.
- 2 Click on the **Configure Palette...** button, then click on the **Node Models** button in the new dialog box.
- 3 Scroll down until you see the server you just created.

- 4 Change the status of the **<initials>_Prod_Features_Server** node model from **not included** to **included**, then click **OK**.

Including the Custom Node Model



- 5 Click **OK** in the Configure Palette dialog box, then save the palette as **<initials>_Prod_Features**. Click **OK** when you are asked to confirm the overwrite.

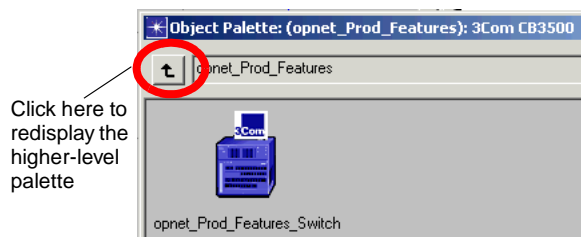
➔ The derived model **<initials>_Prod_Features_Server** appears in the object palette.

Now you can create the third floor subnet, which contains two e-mail servers and the derived 3Com switch. This switch connects to the switches on the other two floors.

- 1 Double-click to enter the **third_floor** subnet.

- 2 Drag two **<initials>_Prod_Features_Servers** from the object palette to the left side of the workspace.
- 3 Now you will add the **<initials>_Prod_Features_switch** to the model. Although an icon with that name is not visible in the object palette, there is an icon named **<3Com CB 3500>**. The brackets around the name indicate that 3Com CB 3500 is not an individual model, but a model group that can include several different models.
 - 3.1 Double-click on the **<3Com CB 3500>** icon to display the individual models in the group (in this case there is only one model in the group).

Object Palette for 3Com CB 3500 Group

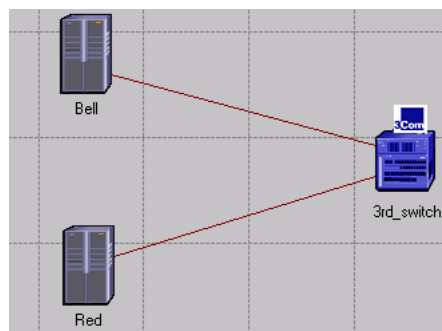


- 3.2 Place an **<initials>_Prod_Features_switch** to the right of the servers.

3.3 Click on the up arrow in the object palette to re-display the higher-level palette.

- 4** Connect the two servers to the switch using **10BaseT** links.
- 5** Using the **Set Name** item from the Object pop-up menu, name one server **Bell**, the other server **Red**, and the switch **3rd_switch**.

The Finished third_floor Subnet

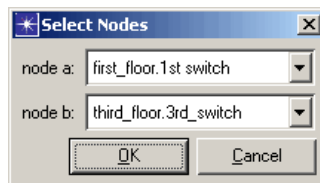


To connect the first and second floors to the third floor:

- 1** Return to the parent subnet by using the Go to Parent Subnet command in the Workspace pop-up menu or by clicking on the toolbar button.

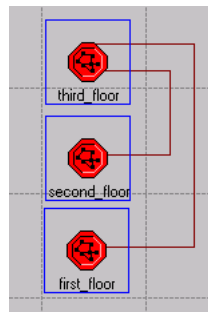
- 2 Connect the **first_floor** subnet to the **third_floor** subnet with a **10BaseT** link. When prompted to select the nodes to connect, select **first_floor.1st switch** and **third_floor.3rd_switch**. Click **OK**.

Select Nodes Dialog Box



- 3 Connect the **second_floor** subnet to the **third_floor** subnet with a **10BaseT** link. Select the **second_floor.2nd switch** and the **third_floor.3rd_switch**. Click **OK**.

Connecting Subnets



- 4 Right-click to turn off link creation.

- 5 Click in the workspace to unselect all objects.
- 6 Save the project.
- 7 Close the object palette.

At this point, your network should be fully functional. However, because this lesson is about productivity tools, we will not run any simulations using the network.

Logical Object Selection

Instead of selecting objects individually, you can use Logical Object Selection to select objects with similar attributes or associations.

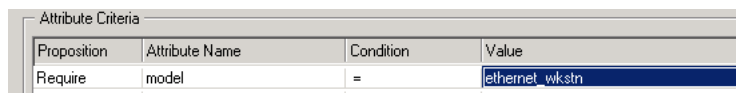
Now that you have created the servers for the network, you should check to make sure that all of the workstation nodes are configured correctly, especially because you don't know how the original model developer set the attributes. Because there are workstations in two different subnets of this model, selecting the nodes one-by-one would be tedious and time-consuming. Instead, you can use Logical Object Selection to select all the workstations at the same time.

To select all the workstations in the network:

- 1 Choose **Edit > Select Objects...**
 - ➔ The Logical Object Selection dialog box opens.
- 2 In the **Object Types** section, use the default settings.

- 3 In the **Include Objects In** section, select **Current subnet**, **Child subnets**, and **Other subnets**.
- 4 In the **Attribute Criteria** section, set the **Proposition** field to **Require**, **Attribute Name** field to **model**, and **Value** to **ethernet_wkstn**.
- 5 Click **OK**.

Logical Object Selection Dialog Box



Proposition	Attribute Name	Condition	Value
Require	model	=	ethernet_wkstn

When you click **OK** in the Logical Object Selection dialog box, IT Guru selects any objects that match the criteria you specified. In this case, IT Guru displays the first subnet found that contains selected objects.

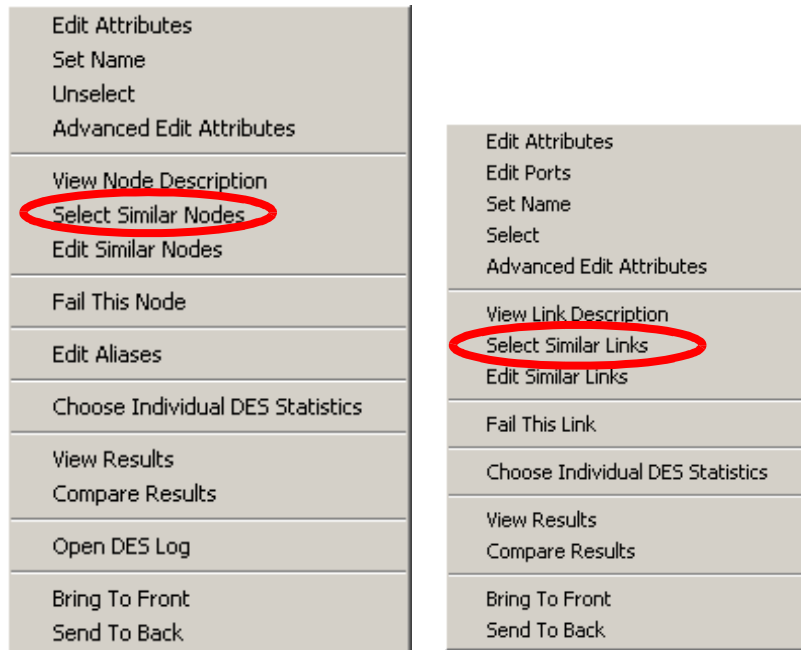
Now that you have selected all the necessary nodes, set their attributes so that 75% of every workstation's CPU is "background-utilized" during a simulation.

- 1 Right-click on any workstation and select **Edit Attributes** from the Object pop-up menu.

- 2 Expand the CPU compound attribute by clicking on the “+”, then change the value of **CPU Background Utilization** to **Edit...**
- 3 In the CPU Background Utilization table, change the **background utilization (%)** attribute to **75** and leave the **time (sec)** attribute at 0.0. Click **OK**.
- 4 Select the **Apply changes to selected objects** checkbox to ensure that the changes you made are propagated to the other selected workstations.
- 5 Click **OK**.
(A message might appear warning you that attribute changes to multiple objects cannot be reverted using the Undo command. Because you are sure you want to make these changes, click **Yes** to accept them.)
 - ➔ The message “**27 objects changed.**” appears in the message area at the bottom of the Project Editor window.
- 6 You can look at the **background utilization** attribute of a workstation in the other subnet to verify that it has been changed.

Another way to select objects in the network is with the **Select Similar Nodes/Select Similar Links** option on the Object pop-up menu. This enables you to select every node or link object in the network whose **model** attribute is the same as the original object. This function is not available on subnet objects.

Select Similar Nodes/Links Option on Object Pop-Up Menu



Network Browser

The Network Browser gives you a tree view and a topology view of the network side by side.

In addition to providing an overview of the network with two views, you can use the Network Browser to traverse the network quickly.

To open the Network Browser:

- 1 Choose **View > Show Network Browser**.
 - ➔ The browser opens with two different views of the network objects: a tree view on the left side of the window and a network topology view on the right side of the window.
- 2 In the tree view, expand the **first_floor** subnet to view a list of its contents.
- 3 Double-click on **1st switch**.
 - ➔ **1st switch** becomes highlighted in the tree view and in the network topology view.
- 4 Right-click on the **1st switch** object (in the tree view or in the network topology view) and select **Edit Attributes** from the pop-up menu.

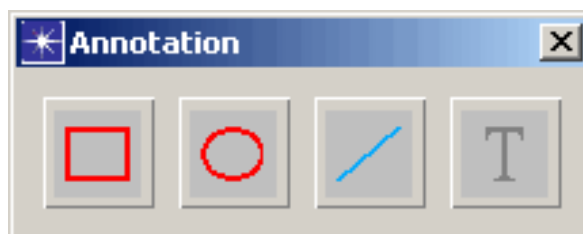
- 5 Change the model attribute to **<initials>_Prod_Features_Switch**, then click **OK**.
 - ➔ This replaces the existing switch with the derived 3Com switch you created with Device Creator.
- 6 Collapse the **first_floor** subnet, then repeat this procedure for the switch on the **second_floor** subnet.
- 7 Choose **View > Show Network Browser** again to close the Network Browser.

Annotation Palette

You can add graphic or text annotations to a network model to improve its readability and understandability. Annotation objects do not affect the functionality of the network model.

The Annotation tool lets you add text, arrows, and other objects to your model to improve readability or to call attention to important features. In this section, you will use the Annotation tool to clarify several points in the building model. Annotation objects do not affect the functionality of a network model.

Annotation Palette



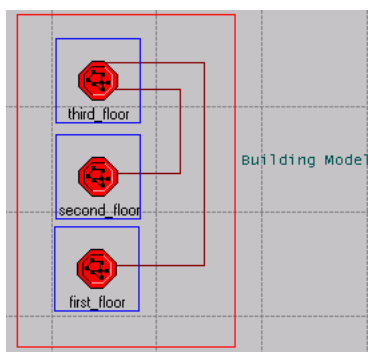
To activate the annotation palette:

- 1 Choose **Topology > Open Annotation Palette**.
 - ➔ The annotation palette opens in the workspace.

Use the annotation tool to show that each subnet represents a floor within a building:

- 1 Traverse the network until the first-, second-, and third-floor networks appear as icons (as shown).

Network Model with Annotations



- 2 Click on the rectangle button in the annotation palette, then draw a rectangle around the three subnets.
- 3 Click on the text button in the palette and type the annotation “**Building Model**” in the text edit pad, then choose **File > Save** from the text edit pad’s menu.
- 4 Specify the location for the text annotation by dragging a box to the right of the rectangle you drew in step 2. Right-click to stop placing text.

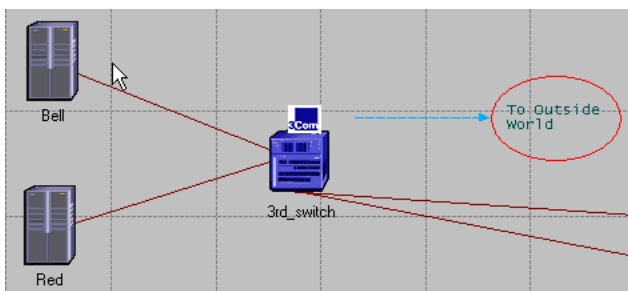
You can also annotate some objects on the third floor to make that part of the model more understandable. Links indicate that the 3rd floor switch is connected to the outside world, but you can add an annotation to make the concept more explicit.

- 1 Double-click on the **third_floor** subnet.
- 2 Add an arrow:
 - 2.1 Click the line button in the palette, then draw a line extending to the right from the switch (left-click at the start point, drag the mouse to draw the line, double-click to end the line at the desired point, then right-click to end line drawing).
 - 2.2 Right-click on the line and select **Edit Attributes**.
 - 2.3 Change the **line style** attribute to **dashed**.
 - 2.4 Change the **head arrow** attribute to **yes** and click **OK**.
- 3 Add the text "**To Outside World**" to the right of the arrow.

- 3.1 Click on the text tool and type the text in the text edit pad.
- 3.2 Save the text.
- 3.3 Drag the cursor to place the text in the workspace. (Right-click to stop placing text.)
- 3.4 Draw an ellipse around the text.

The third floor subnet should now look like this:

Third Floor Subnet with Annotations



You can add more annotations to the network model if you want. You can also change the color of any annotation object by changing its **color** attribute. To hide all annotations, choose **View > Annotations > Show in Subnet**.

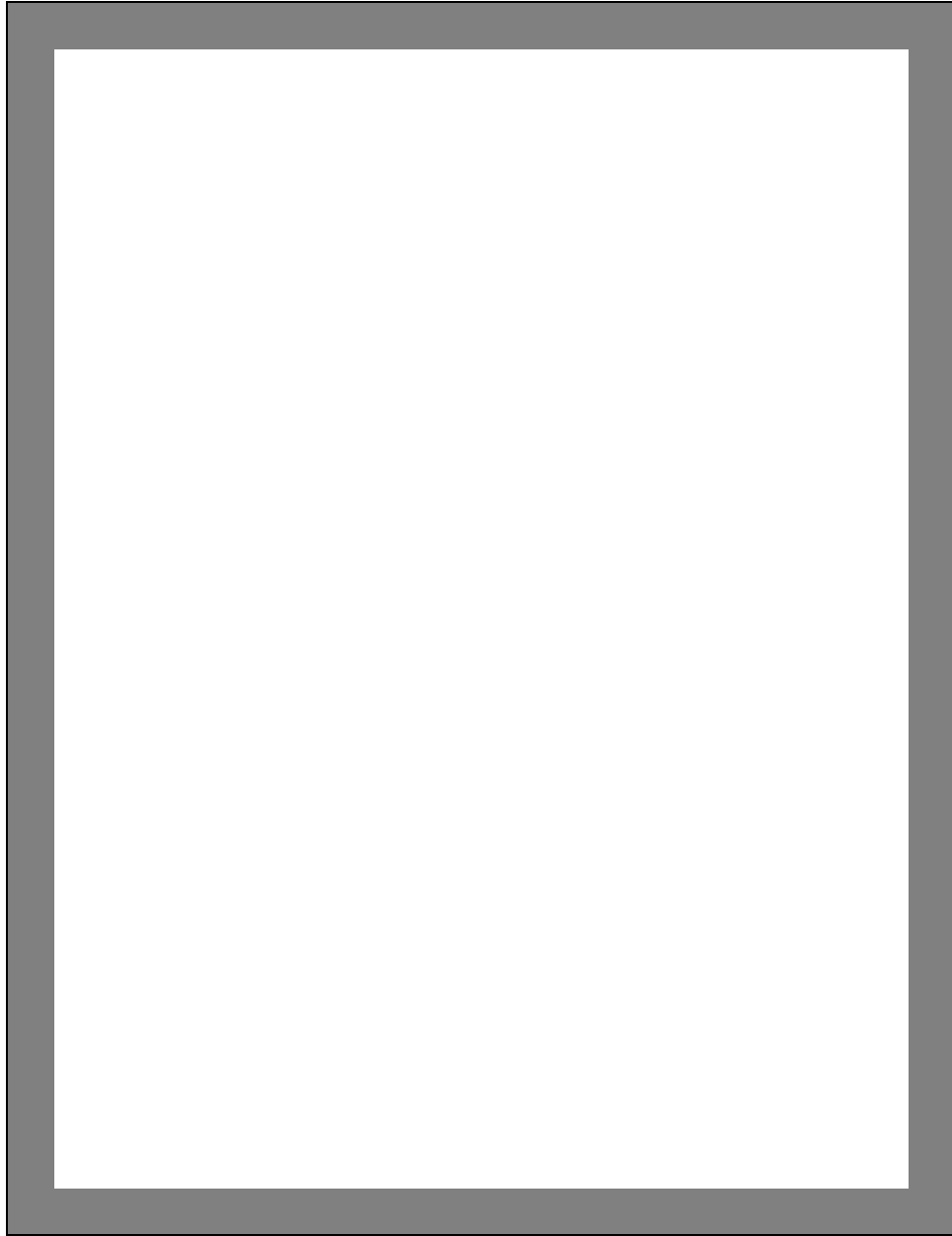
When you finish adding annotations to your model, save it.

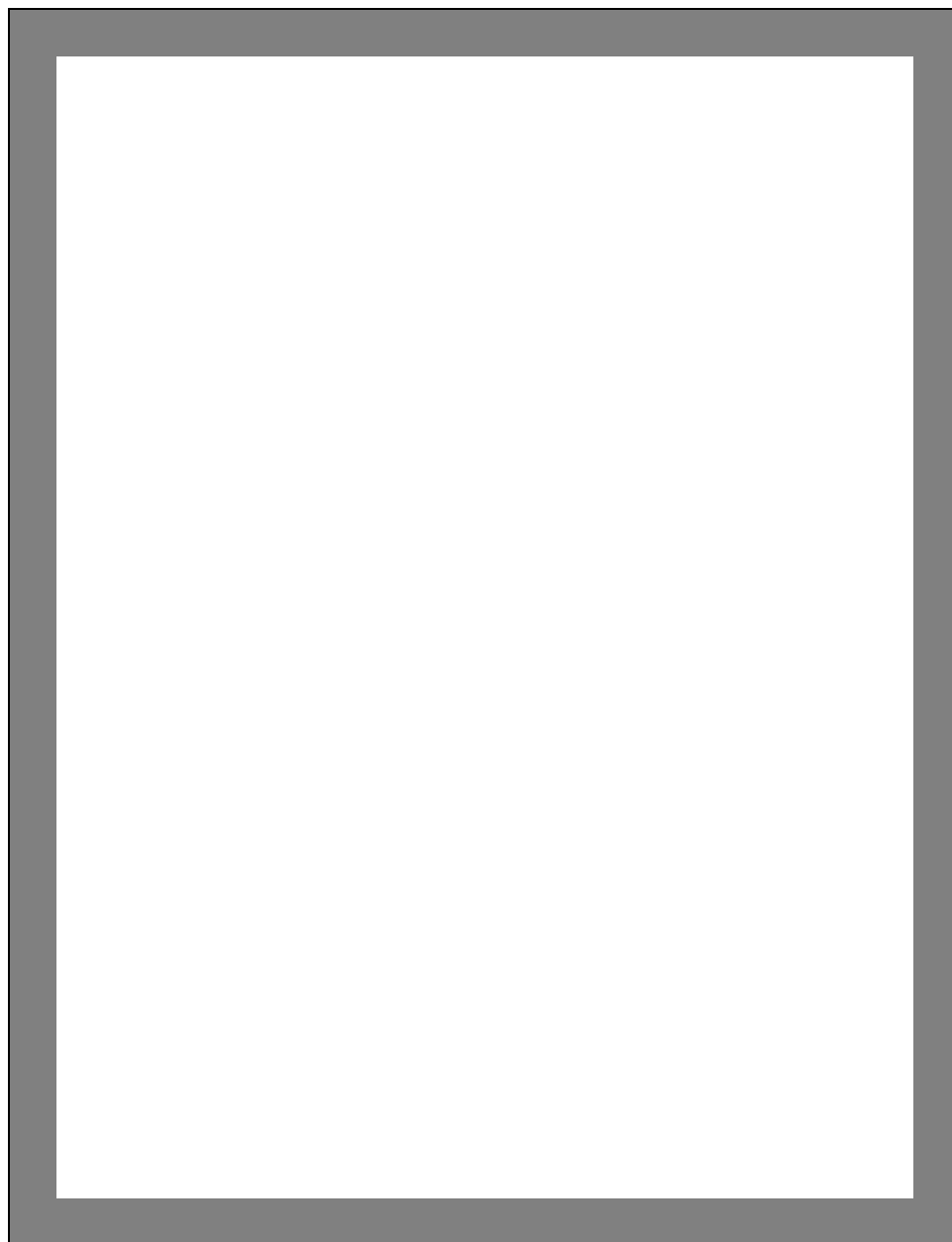
Congratulations! You have completed the core tutorial lessons. Now you can build your own network model, collect statistics, run a simulation, and analyze the results.

If you purchased additional modules, such as Multi-Vendor Import or ACE, continue with the tutorials that illustrate these capabilities. Return to the main tutorial menu and choose the desired tutorial from the list of available lessons.

From time to time, you might have questions about certain aspects of IT Guru. For answers, see **Help > Product Documentation** or visit the OPNET Technical Support website by choosing **Help > Web - Support Center**.

Good luck model building!





App A Troubleshooting Tutorials

Troubleshooting IT Guru Tutorials

Introduction

This document contains information that can help you correct common problems.

- [The OPNET Installation Directory](#)
- [The tutorial_req Files](#)
- [Saving Tutorial Files](#)
- [The tutorial_ref Directory](#)
- [Troubleshooting Simulations](#)
- [Additional Help](#)

The OPNET Installation Directory

To complete the tutorials, you must install the IT Guru standard models, which include the required tutorial models. The standard models are normally installed when you install the Standard Model Library.

IT Guru standard models apply to common protocols and vendor devices. The standard models are in the subdirectories under the OPNET release directory (<reldir>):

<reldir>\models\std*protocol_name*

<reldir> describes the directory that contains the current IT Guru software.

You can find your **<reldir>** by performing the following steps:

- 1 In the main menu, select **Help > About This Application**.
- 2 In the About IT Guru dialog box, click on the **Environment** tab, and then expand the **System Information** section.
- 3 Under System Information, find the **OPNET release directory**.

For example, the **<reldir>** for a default installation of this release of IT Guru on Windows is

C:\Program Files\OPNET\11.0.A

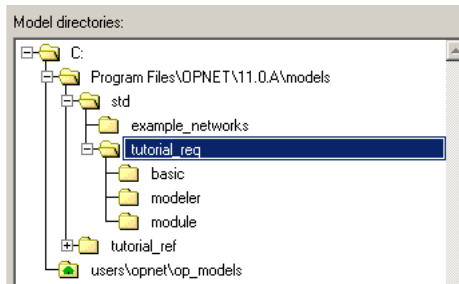
The tutorials use the Windows convention of the backslash character (\) as the separator in directory pathnames. If you are using Solaris, replace the backslash with a forward slash (/).

The tutorial_req Files

When you perform a tutorial, you are asked to open specific tutorial model files. These model files are *required* to complete the tutorial; they are located in the **<reldir>\models\std\tutorial_req** directory or its subdirectories.

When you are asked to open supplied model files, make sure you navigate the directory structure in OPNET's open file browser (in the left pane) to the **tutorial_req** directory.

Navigating to the tutorial_req Directory

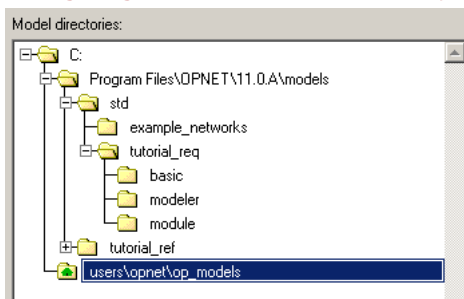


Saving Tutorial Files

In several tutorials, you will be asked to open an OPNET model file and save it with a unique prefix, such as your initials (<your_initials>). This way, several users can create and complete their own copy of the same tutorial without interfering with each others' work.

When you save a new model file, or use the **Save As...** command to save an edited model file with your initials, remember to navigate to your OPNET default model directory (normally **<home>\op_models**) to save your files.

Navigating to the op_models Directory



If you do not navigate to your default model directory (normally **<home>lop_models**) when you perform a **Save As...** operation, the file is automatically saved in the current model directory.

The tutorial_ref Directory

The **tutorial_ref** directory contains completed *reference* models for many tutorials.

If you are unable to complete a tutorial lesson, you can compare your model to the **tutorial_ref** model to help you determine the cause of the problem.

The tutorial_ref directory for IT Guru is

<reldir>\models\tutorial_ref\<subdirectory>

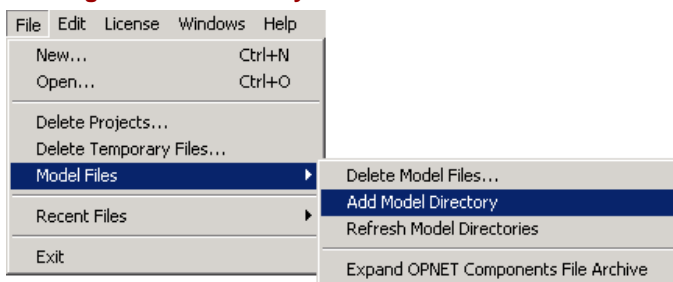
The subdirectory contains models according to the category under which they are listed on the Tutorials menu for your product (for example, basic or module).

If the tutorial_ref directory is not available in the browser, use the following procedure to add the tutorial_ref directory to your model directories.

To add the **tutorial_ref** directory to your model directories list, perform the following steps:

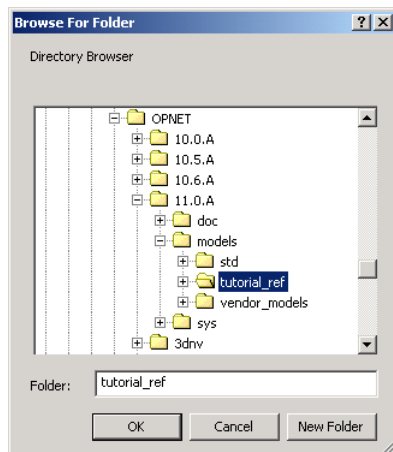
- 1 Select **File > Model Files > Add Model Directory**.

Adding a Model Directory



➔ The Browse For Folder dialog box opens.

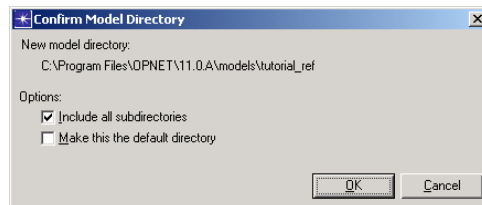
Locating the tutorial_ref Directory



- 2 In the Browse For Folder dialog box, navigate the directory tree to the **tutorial_ref** directory, choose the directory, and then click **OK** (or **Choose** in Solaris).

➔ The Confirm Model Directory dialog box opens.

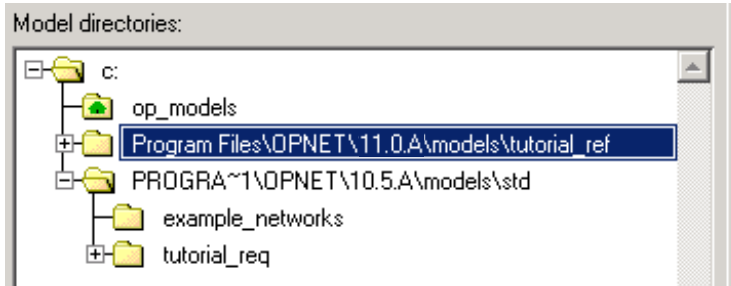
Adding a Model Directory



- 3 Select the **Include all subdirectories** checkbox, and then click **OK**.

➔ The **tutorial_ref** directory displays under Model directories (and is also included in your mod_dirs Preferences) making the reference models available to you.

The tutorial_ref Directory and Its Subdirectories



Troubleshooting Simulations

If your simulation did not complete as expected, you will need to discover the problem, which is usually with your project or with the product installation.

If a simulation does not run successfully with your model, but does run successfully using the **tutorial_ref** model, you can assume that the problem is with your model.

Run the simulation in the **<tutorial_lesson>_ref** model:

- 1 Select **File > Open....**
- 2 Navigate to the correct folder in the **tutorial_ref** directory, and then click on the **<tutorial_lesson>_ref** model. One example of a completed tutorial can be found at **tutorial_ref/basic/Sm_Int_ref**.
- 3 Select a scenario and then run the simulation.

If the simulation runs successfully, the problem is in your model.

Start with the first page of the tutorial and carefully compare the instructions with your model configuration.

You can also compare your model with the **<tutorial_lesson>_ref** model.

Additional Help

If a simulation or analysis does not complete, the problem might be with the installation.

- 1 Read the installation instructions on the CD case.
- 2 Check the Website for recent updates to the tutorials, models, and product installation instructions (www.opnet.com/support).
- 3 Check the FAQ section of the OPNET Website (www.opnet.com/support).

