Importing Traffic

Overview

This lesson demonstrates the traffic import capabilities of Modeler. 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 a topology and 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>\OPNET\10.0.A\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**, and rename the project **<your_initials>_Imp_Data**.

There are several types of traffic in Modeler:

- **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 Device/Link 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 baseline 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.

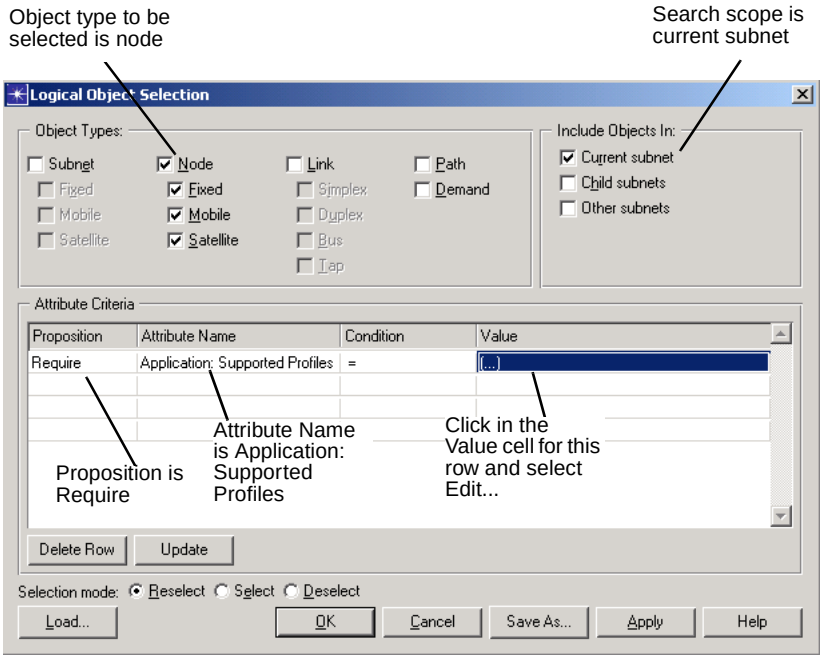
Verify that the **FTP (Light)** profile is assigned to the LAN objects and that the application **File Transfer (Light)** is supported on the LAN server.

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



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

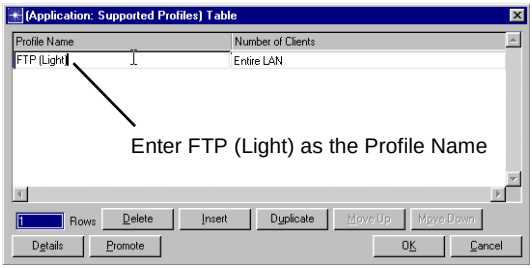
3.1 Click in the **Value** cell of the first row in the Attribute Criteria table and select **Edit...**

➥ 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) Table Dialog Box



3.4 Click **OK** to close the **(Application: Supported Profiles) Table** 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) Table** 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.

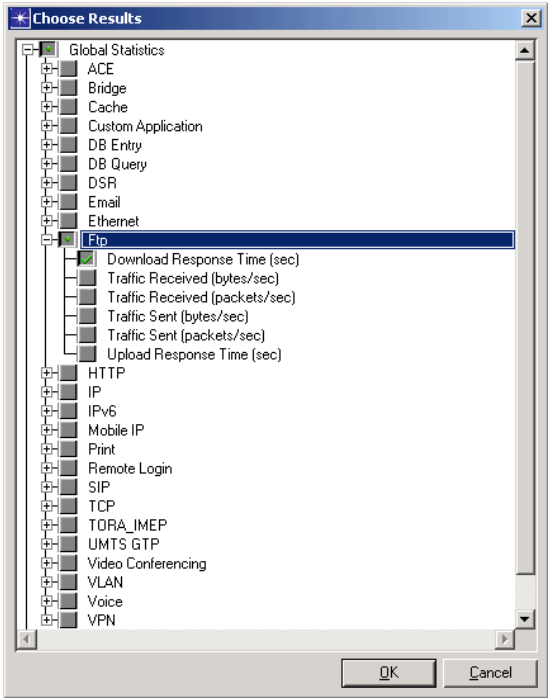
   ➥ Notice the green dot in the checkboxes next to **Global Statistics** and **Link Statistics**. This indicates that statistics in these hierarchies have been chosen for collection.

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 Modeler 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 Modeler 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 Modeler software. You can find your **<reldir>** directory by doing the following:

1. Select **Help > About This Application** in the main menu.

   ➥ The About OPNET Modeler 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 Modeler 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 Modeler 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 Modeler and applied to the network in the current scenario.

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

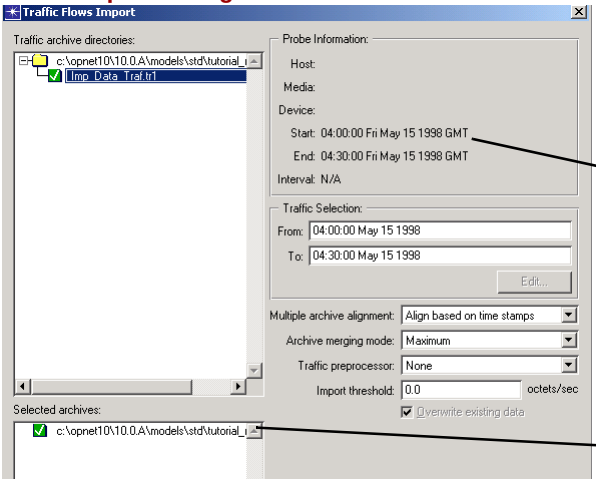
   ➥ The Traffic Import dialog box appears.

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.

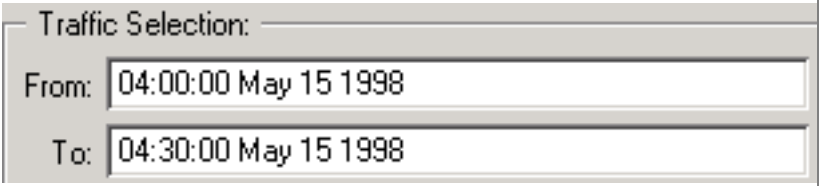
Traffic Import Dialog Box



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



Another attribute that can be set during import is the **Archive Merging Mode**. This specifies how Modeler 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

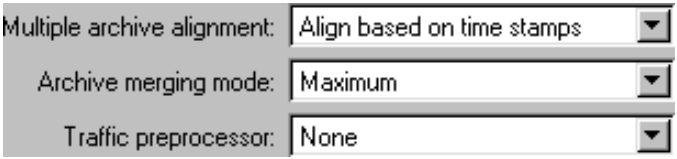
In some cases, the **addition** mode is the correct choice. Say, for example, a flow from node 1 to node 2 is split into pieces and sent on multiple paths (one through node A, another through node B).

- 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



2 Click **Import**.

➥ 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, Modeler 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, Modeler matches imported traffic with endpoints in the network using the node's network alias (usually the node's IP address).

Sometimes Modeler 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

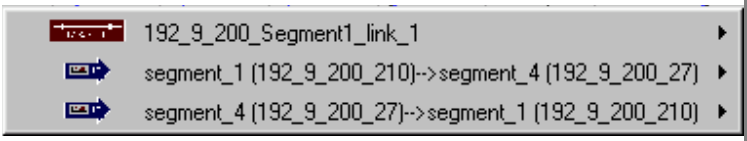
In this section we will examine the traffic you just imported. In the process, you will learn how OPNET models end-to-end background traffic using traffic flow objects.

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.

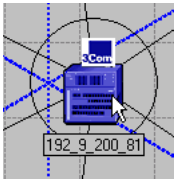
➥ A popup menu appears and displays three different objects: the link object (brown rectangle) and two traffic flow objects (blue arrows).



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 may be hidden by other objects. To view all objects at a particular point in the network, right-click and examine the pop-up menu.

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

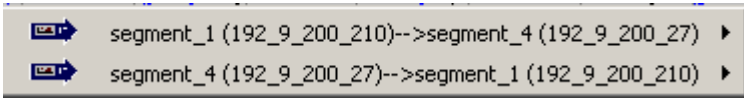
3 Click on the center switch (192_9_200_81) to select it; then move this switch so that the blue arrow between the segment_1 LAN (upper right) and segment_4 LAN (lower left) is visible.



➥ The link (solid black arrow) between the switch and segment_1 link that was hidden by the segment_1 <--> segment_4 traffic flows (blue dashed arrows) becomes visible.

4 Right-click on the blue arrow between the segment_1 LAN and the segment_4 LAN.

➥ A popup menu appears and displays two different traffic flow objects. One specifies traffic from segment_1 to segment_4, and the other specifies traffic from segment_4 to segment_1.



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.)

5 Click in the project workspace.

6 Open the Edit Attributes dialog box for the segment_1 --> segment_4 traffic flow:

6.1 Right-click on the traffic flow between the segment_1 and segment_4 LAN.

UNIX: Do not release the right mouse button.

6.2 Select the segment1-->segment4 traffic flow. This opens the context-sensitive menu for this object.

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

➥ The Edit Attributes dialog box for this traffic flow opens.

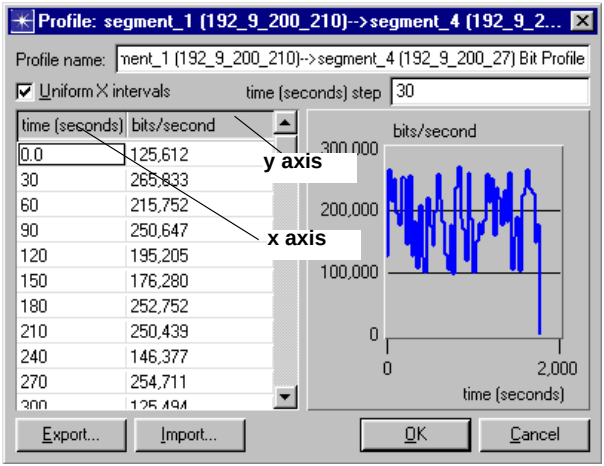
The model (ip_traffic_flow) used to create this flow object has two attributes that specify the source-to-destination traffic:

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

These attributes describe the traffic data you just imported.

7 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.

8 Click **OK** to close the **Profile** dialog box, then **OK** to close the Edit 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.

Modeler can be used to model organizational scaling quickly and easily.

First, you need a baseline.

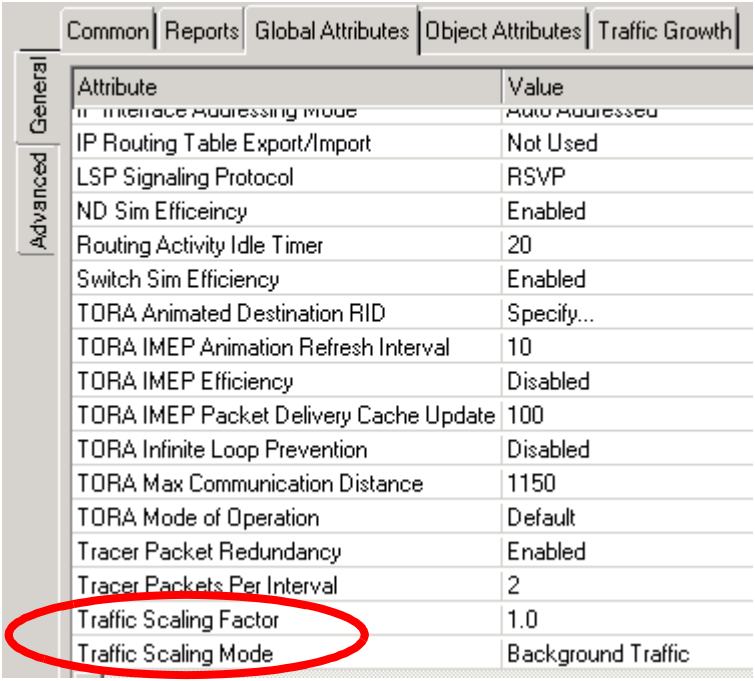
Run the Simulation

1. Choose **DES** > **Configure/Run Discrete Event Simulation**.

2. Click on the **Common** tab and verify that the **Duration** is 360 seconds.

3. Click on the **Global Attributes** tab and verify that
   - The **Traffic Scaling Factor** is 1.0
   - The **Traffic Scaling Mode** is Background Traffic

Configure Discrete Event Simulation Dialog Box



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 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...**

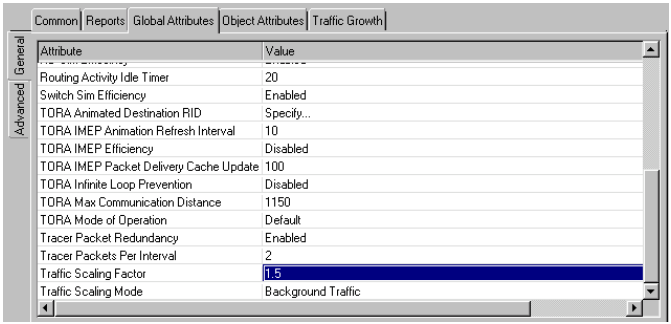
2 Name the new scenario **scaled_50_percent**.

3 In the new scenario, choose **DES** > **Configure/Run Discrete Event Simulation**.

3.1 Click the **Global Attributes** tab 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 device/link baseline load traffic, but not explicitly generated traffic).

Traffic Scaling Factor is Set to 1.5



The next time you run a discrete event simulation, the background traffic in the network will be increased by 50 percent.

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.

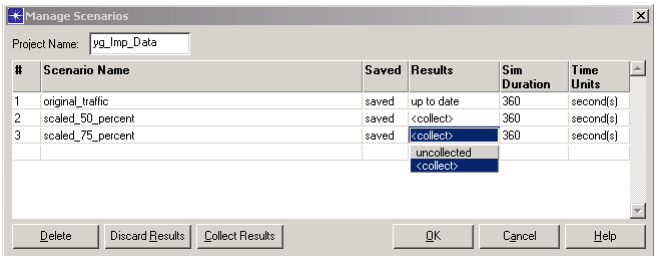
Now that each scenario has been created with the appropriately scaled traffic, you can run the two new scenarios as a batch.

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 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**

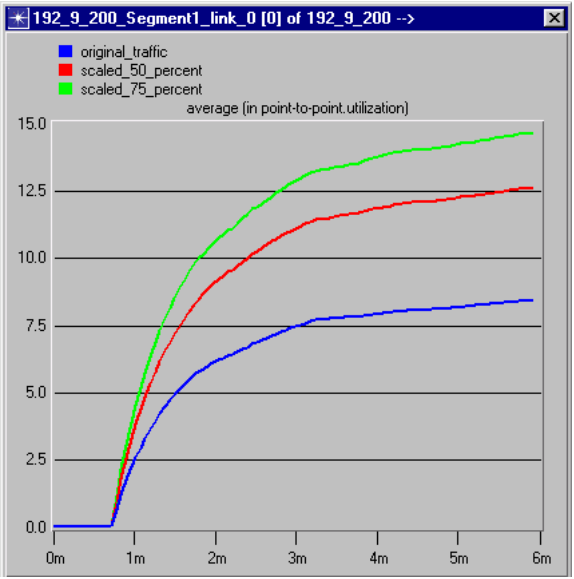
Compare Results Dialog Box (detail)

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 50 percent for the **scaled_75_percent** scenario) indicates that the between-LAN links are not the bottleneck in this network.

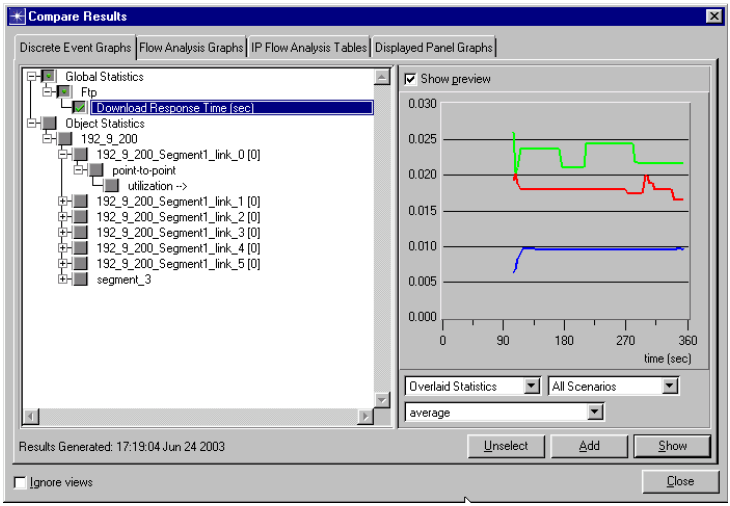
The other statistic you collected was Download Response Time:

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

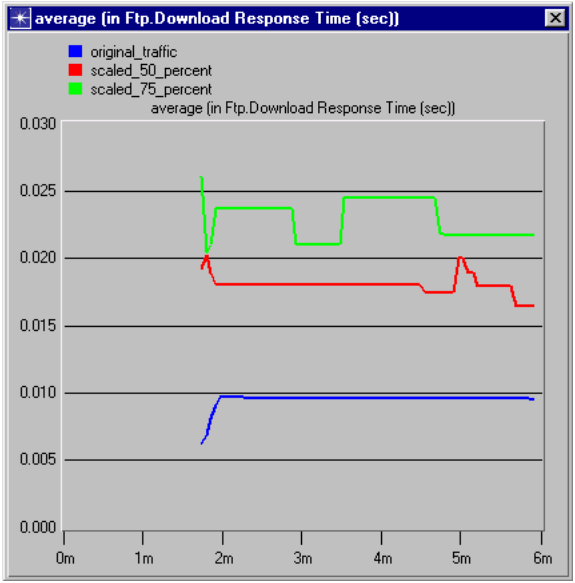


4. Verify that the graph filter is set to **average**.

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 choose another tutorial as follows:
  - If you have ServiceProvider Guru, continue with the Expert Service Prediction (ESP) tutorial.
  - If you have IT Guru with the optional ESP module installed, continue with the ESP tutorial. If you do not have the ESP module installed, continue with the Productivity Features tutorial.