

# ***Small Internetworks***

---

## **Introduction**

In this lesson, you will see how Modeler can do organizational scaling to solve a typical “what if” problem. You will learn how to use Modeler features to build and analyze network models. This lesson focuses on the use of the Project Editor, and how it will be used with the Node and Process editors in later lessons. In this lesson, you will

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

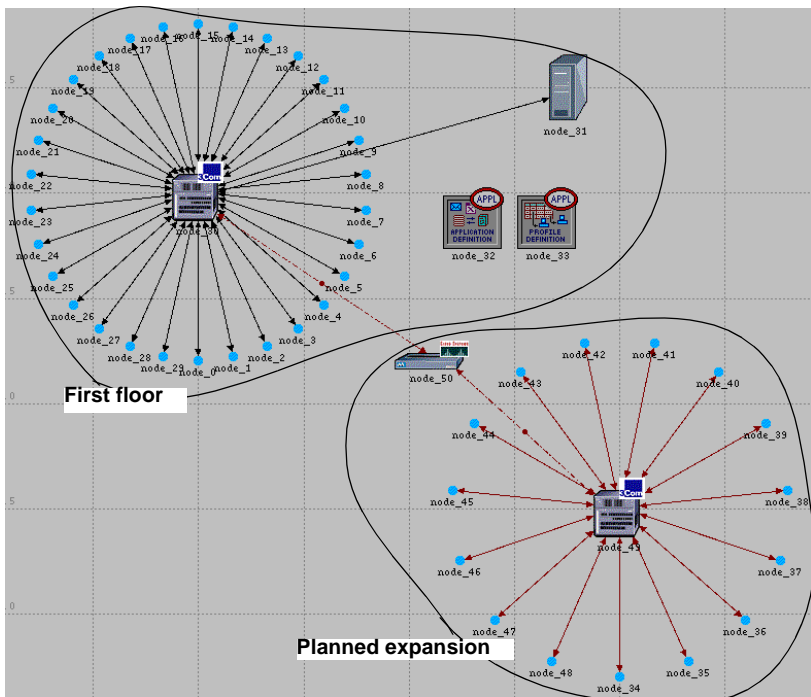
---

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

---

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

---

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

- 1 If Modeler 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**

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

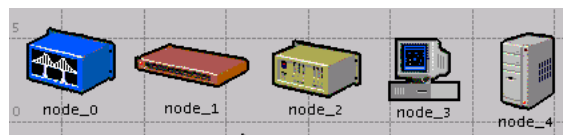
- ➔ A workspace of the size you specified is created. The object palette you specified opens in a separate window.

## 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.

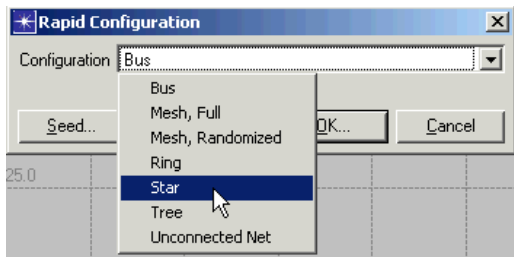
You can use any of three methods to create a network topology, or a combination of all three. One method is to import the topology (discussed in a later lesson). Another is to place individual nodes from the object palette into the workspace. The third method is to use **Rapid Configuration**.

**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 drop-down menu of available configurations, then click **OK...**

#### Available Configurations Pop-up Menu



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

**<protocol1>\_...\_<protocoln>\_<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**).

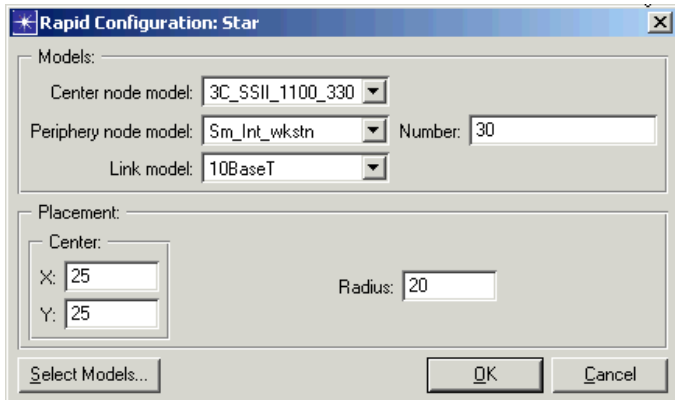
To specify the nodes and links to use to build the network:

- 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 provides 30 Ethernet workstations as the peripheral nodes.
- 3 Set the **Link Model** to **10BaseT**.

Specify where the new network will be placed:

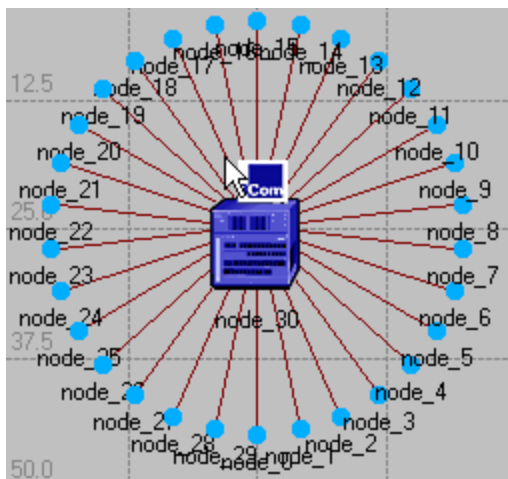
- 1 Set the **X center** and **Y center** to **25**.
- 2 Set the **Radius** to **20**.

### 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:

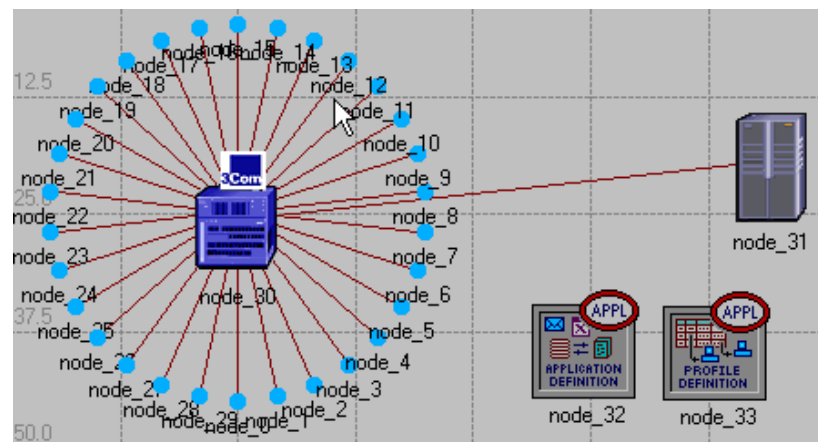
- an application definition object with the default configurations of the standard applications, and
- a profile definition object with a profile that models light database access

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.

However, first let's explore the Node and Process Editors.

---

The Node and Process Editors are integral to the OPNET workflow. The Node Editor is used to create node models that describe the internal flow of data within a network object. The Process Editor is used to create process models that describe the behavioral logic of a module in a node model.

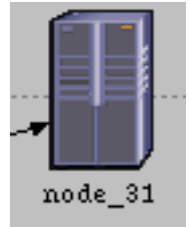
---

Every network object (except links) has an underlying node model that specifies the internal flow of information in the object. Node models are made up of one or more **modules** connected by **packet streams** or **statistic wires**. Node modules in turn contain process models. A process model is represented by a **state transition diagram (STD)** that describes the behavior of a node module in terms of **states** and **transitions**.



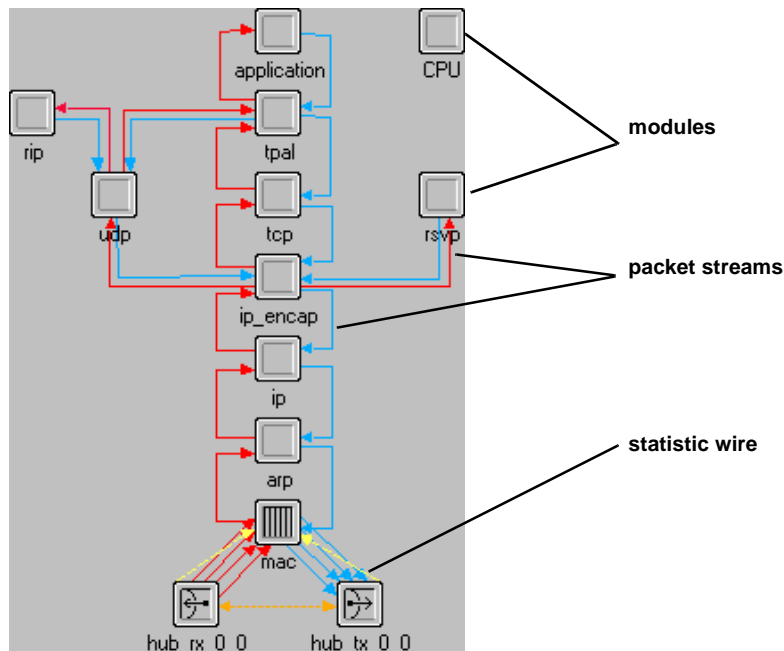
Let's explore the node model that controls the server in the first floor network:

- 1 Double-click on **node\_31** (the **Server object**) in the Project Editor.
  - ➔ The Node Editor opens as a new window within Modeler.



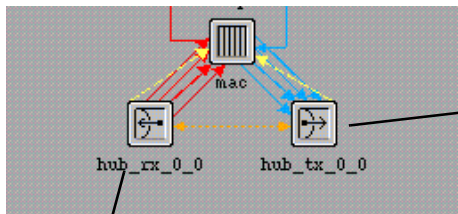
The following figure shows the node model within the Ethernet Server network object. The node model is made up of several different types of **modules** which are described in the M/M/1 tutorial. **Packet streams** and **statistic wires** connect the modules.

## Ethernet Server Node Model



During a simulation, packets sent from a client machine are received by the hub receiver object (`hub_rx_0_0`) and processed up the protocol stack to the application module. After processing, they are sent down the stack to the transmitter (`hub_tx_0_0`), then back to the client machine.

## Packet Processing by the Node Model

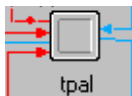


After processing, packets are sent by the hub transmitter object

Packets are received by the hub receiver object

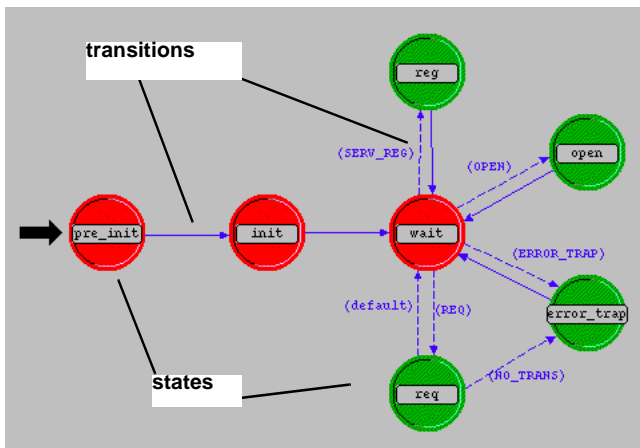
Next, let's look at the process model that defines the behavior of the **tpal** module. To view the process model:

- 1 Double-click on the **tpal** module in the Node Editor.



- ➔ The Process Model Editor opens in a new window.

## Example Process Model



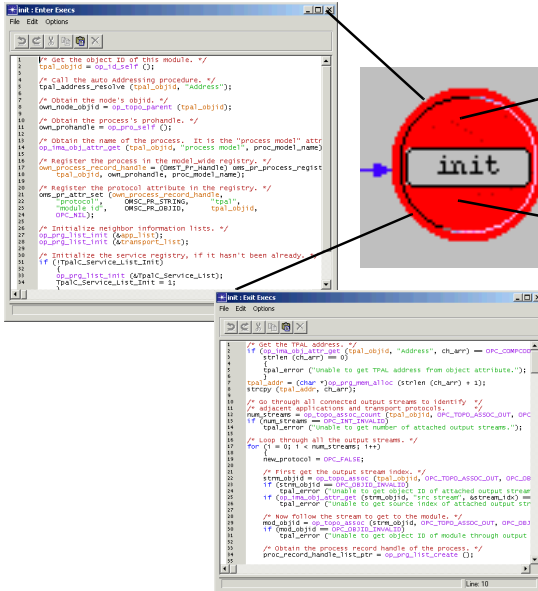
- Note the red and green states (these will be discussed in greater detail in the Basic Processes lesson) and the solid and dotted lines indicating transitions between the states.

Each state in the process model contains an **enter executive** and an **exit executive**. Enter executives are executed when a process enters a state. Exit executives are executed when the process leaves the state. Operations performed in the state are described in C or C++.

- Open an enter exec by double-clicking on the top half of the **init** state.

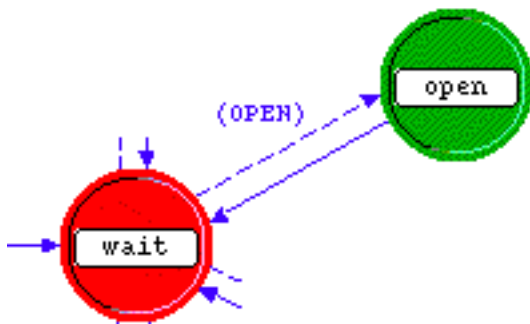
- Open an exit exec by double-clicking on the bottom half of the state.

## Opening the Enter Exec or Exit Exec of a State



the transition can occur. However, the transition from **open** to **wait** (solid line) is unconditional, and will occur whenever the code in the **open** state has finished execution.

### Conditional and Unconditional Transitions



Later lessons explore these editors in greater depth.

- 6 Close the Node and Process editors. If prompted, do not save changes.

## 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.

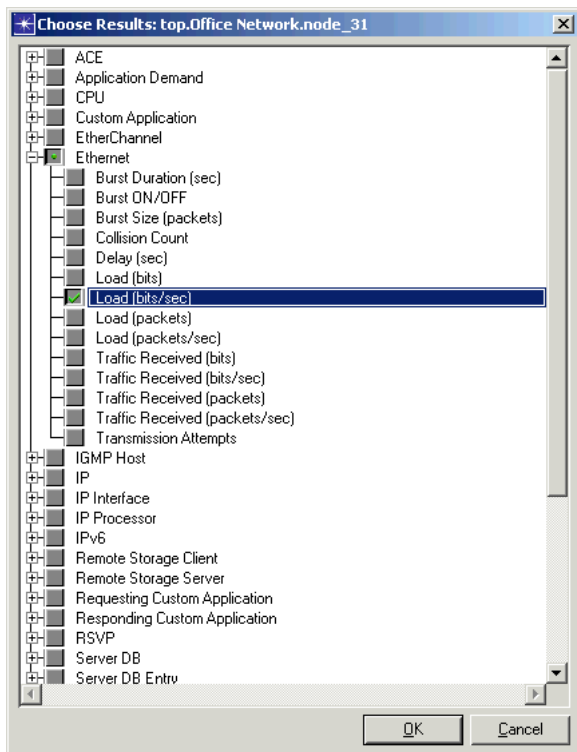
↳ The Choose Results dialog box for node\_31 appears.

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

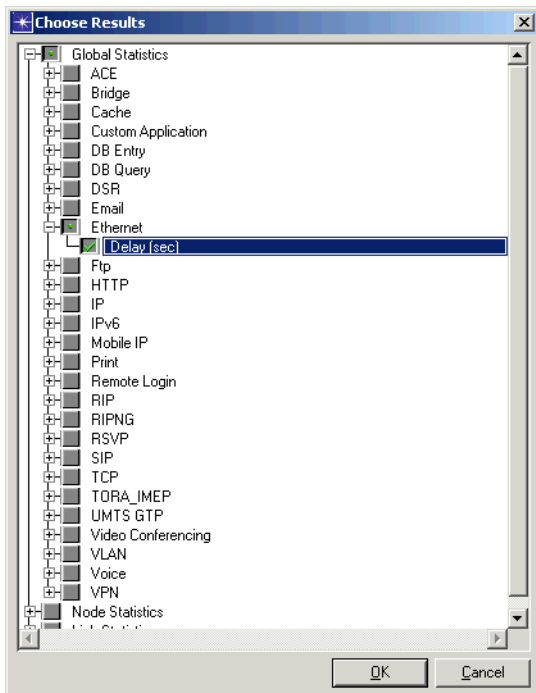


- 3 Click the checkbox next to **Load (bits/sec)** to enable collection for that statistic.
- 4 Click **OK** to close the dialog box.

**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:

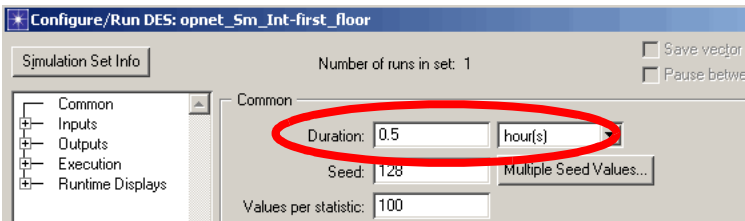
- 1 Select **DES > Configure/Run Discrete Event Simulation....**

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



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

### Configure Discrete Event Simulation Dialog Box



### 3 Set the Simulation Kernel to optimized.

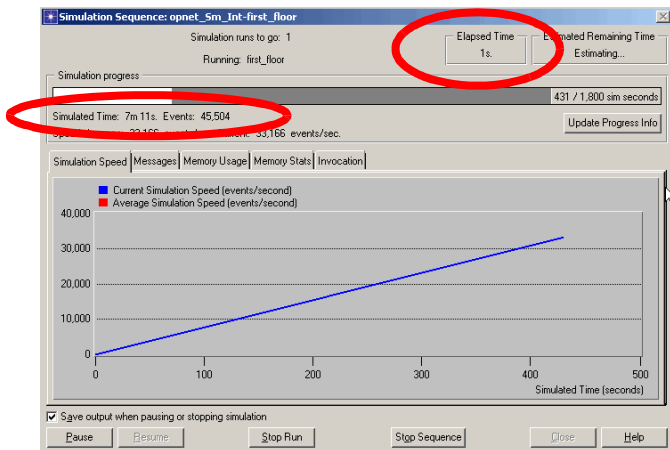
You can use one of two types of kernels to run your simulation. The development kernel collects simulation data you can use to debug your models, but the optimized kernel runs faster.

### 4 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 2 seconds of elapsed (actual) time, Modeler has simulated 4 minutes and 29 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.

- 5 When the simulation finishes, the contents of the Messages tab appears. Click the **Close** button in the Simulation Sequence dialog box.
- 6 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 Modeler 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.

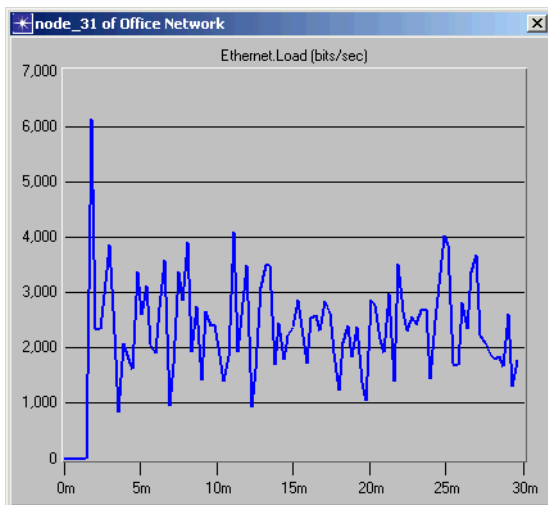
To view the server Ethernet load for the simulation:

- 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.

- 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

Simulation time, in minutes



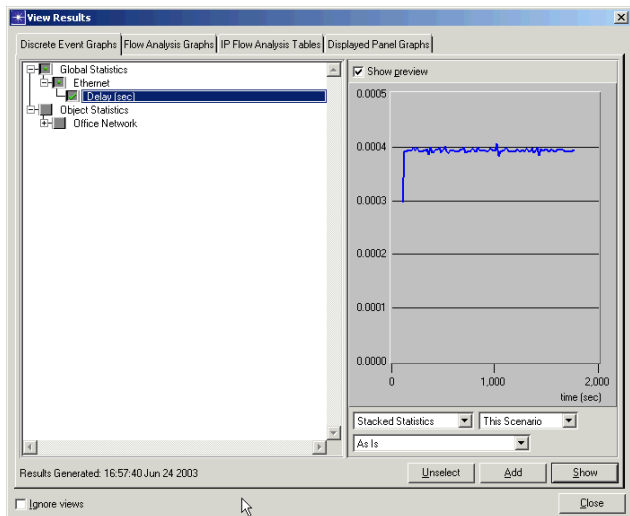
Note that at its peak, the load on the server is about 6,000 bits/second. You will need this baseline for comparison after you add the second network.

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.

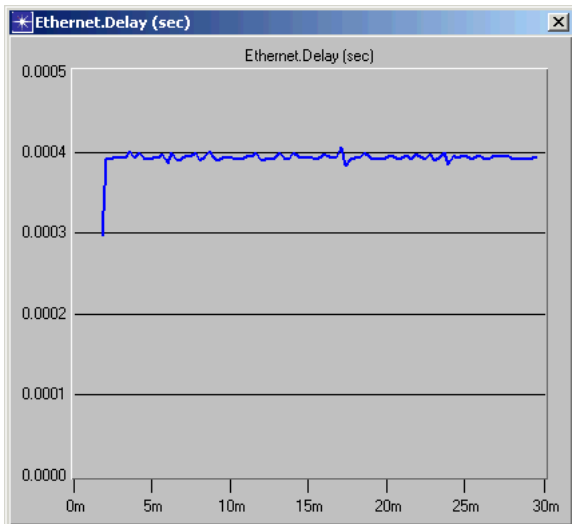
## Viewing Ethernet Delay for the Whole Network



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

## Ethernet Delay Graph

Seconds



Simulation time, in minutes

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.

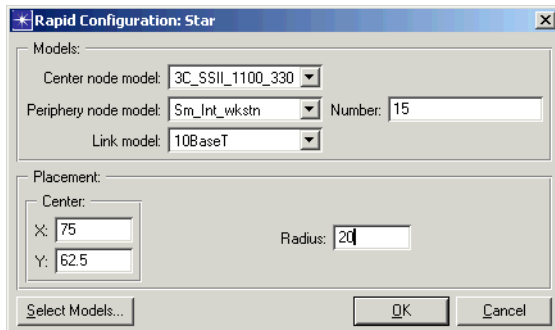
To duplicate a scenario:

- 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



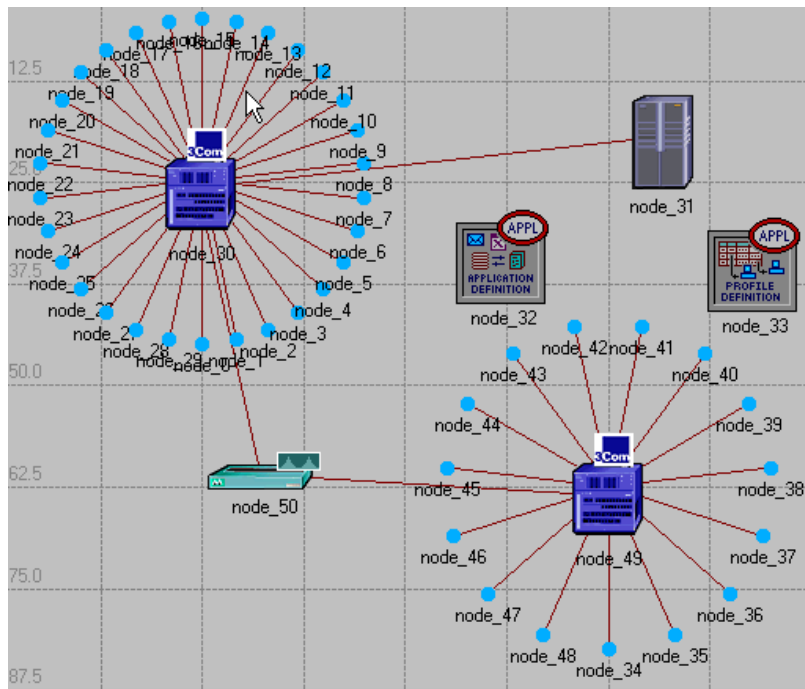
- 4 Click **OK** to create the network.

Join the two networks:

- 1 If it is not already open, click the toolbar button to open the object palette. The image shows a square toolbar icon with a light gray background and a thin black border. Inside the square, there are several small, colorful icons: a yellow triangle, a red heart, a blue square, a green circle, a black pencil, a red eraser, and a red arrow pointing upwards.
- 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

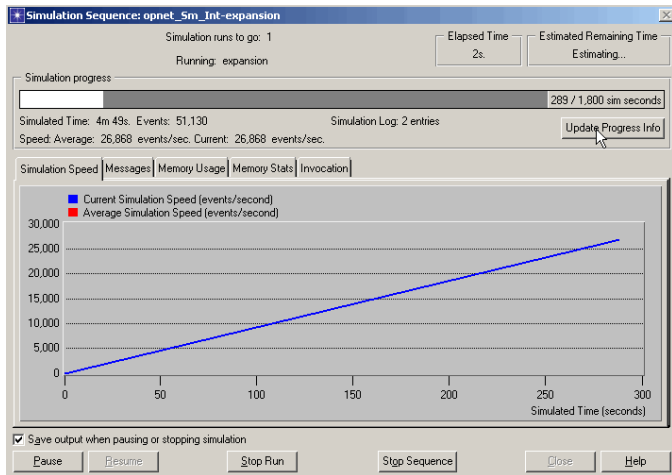


To run the expansion scenario:

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

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

### Simulation Sequence Dialog Box, Simulation Speed Tab Selected



- As before, a window appears showing the simulation's progress. When the Simulation Speed tab is selected, an animated graph shows both the current and average speed in events per second.
- When the simulation is done, close the Simulation Sequence dialog box. If you had problems, see ["Troubleshooting Modeler 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 View Results dialog box appears containing a hierarchical representation of statistics collected for the server node.

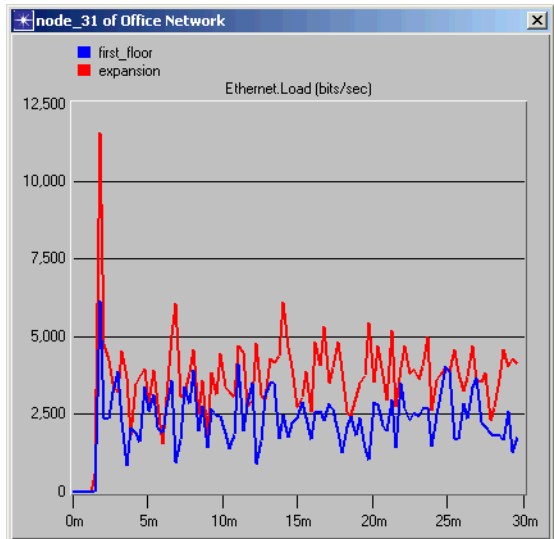
If your results differ radically from those shown in the following figures, you will have to troubleshoot your simulation. See "[Troubleshooting Modeler Tutorials](#)".

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

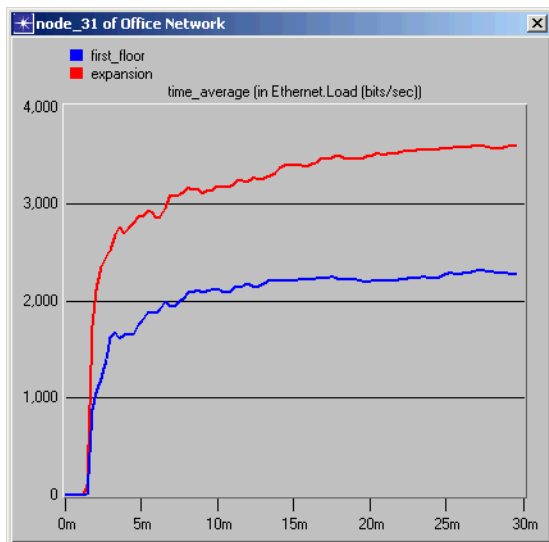
bits/second



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.

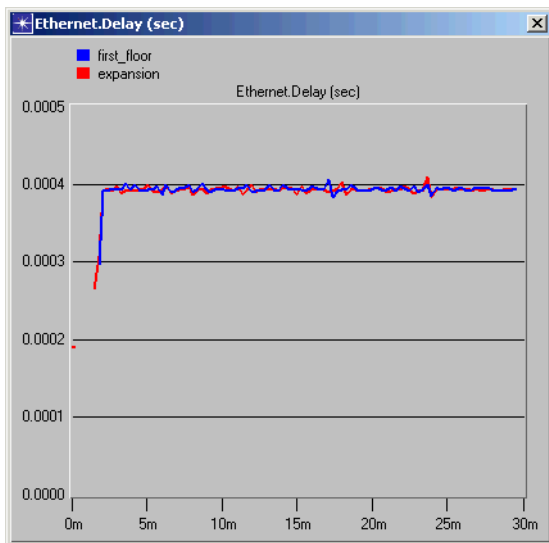
The last step is to see how much the network's delay is affected by adding a second floor. To compare Ethernet delay for the two scenarios:

- 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

Seconds



Simulation time, in minutes

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 M/M/1 Queue tutorial. This lesson explores the use of node models in the OPNET workflow. Return to the main tutorial menu and choose **M/M/1 Queue** from the list of available lessons.