# Simulate the Topology

Simulate the Topology Overview

The simulation phase is when you run the simulation of your topology design. The **Simulation** perspective provides a set of views that support the simulation phase. By comparison, the design and build activities occur in the **Design** perspective, which provides a set of views that support the design activity. Some views in the **Simulation** perspective can also be viewed in the **Design** perspective.

Simulation Perspectives and Views

The main areas of focus within the **Simulation** perspective are the **Simulations** view and the **Console** view. The following figure highlights the **Simulation** and **Console** views for a running simulation.

Figure 1: Simulation Perspective Initial Display
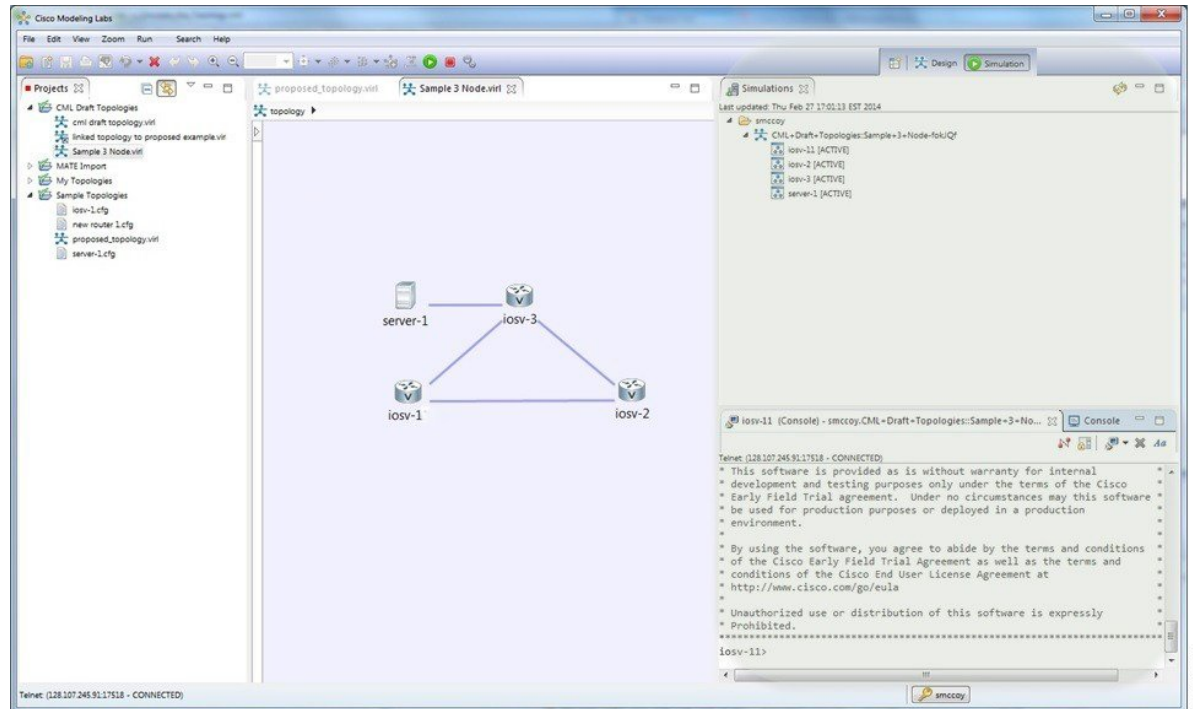
From the **View** menu, you can open additional views in the perspective and arrange the open views by dragging them within the perspective workspace. See the online help and the section Customizing Perspectives for more information.

Note   To reset your current perspective to its original configuration when the workbench was first opened, right-click the perspective button and select **Reset**.

Node Limits

In Cisco Modeling Labs Corporate Edition, the number of nodes you can run is limited by the resources available on the server and the number of licenses available. In Cisco Modeling Labs Corporate Edition, the design limit is 1,000 nodes.

During the simulation phase, you can perform the tasks described in the following sections.

Launch a Simulation

Before You Begin

- Complete the topology design.

- Complete the task of building the nodes and interfaces.

- (Optional) Generate the configuration using AutoNetkit.

⚠️

Caution   When you manually make changes to a node configuration and bypass AutoNetkit auto-generation, those changes do not appear in the topology view of the **Design** perspective or **Simulation** perspective. For example, if you use the **hostname** command to change the host name from `iosv-1` to `Router-1` in the configuration, the node name in the topology view and in other related views remains as `iosv-1`.

- Open the desired topology.

✎

Note   The topology should be open and visible in the **Topology Editor**. If you have multiple topologies open in the **Topology Editor**, simulation will launch for the currently active view.
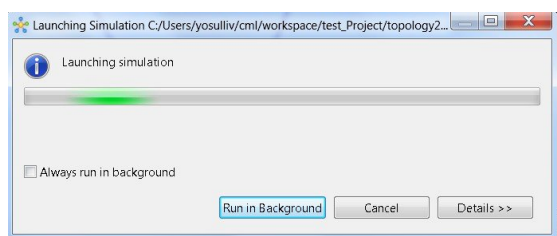
Step 1   From the toolbar, click the **Launch Simulation** button.
The simulation launches and provides a unique identifier, which means that multiple instances of the topology can be launched and each will have a unique name.

**Step 2**   In the **Launching Simulation** dialog box, select any of the following actions:

a) Check the **Always run in the background** check box. All future node start requests, stop requests, and simulation launch requests run in the background and do not display dialog boxes.
   Note   To control the background setting, choose **File** > **Preferences** > **General**.

b) Click **Run in background**. The dialog box closes when the node simulation starts.

c) Click **Cancel** to return to the **Design** view.

d) Take no action, and the node simulation launches momentarily.
   Tip   When you click **Run in background**, the status bar displays a progress icon. Click the icon to display a compact view of the activity progress. If the background activity encounters an error, the icon shows a red **X**. Click the error icon to display the error dialog.

Step 3   In the **Simulation launched** dialog box, click **OK**.
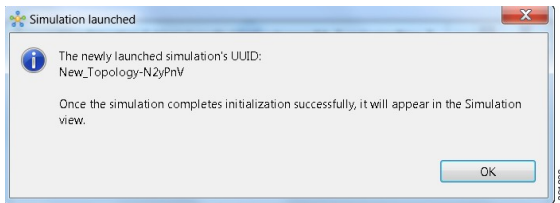
**Step 4**   When prompted to switch to the **Simulation** perspective, click **Yes**.
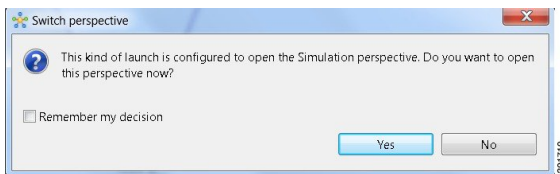
> **Note**   We recommend that you switch to the **Simulation** perspective to view the running simulation and to connect to node consoles.

The Cisco Modeling Labs client tracks the state of the simulations. All launched simulations appear in the **Simulations** view. Status messages are displayed in the **Console** view. After the Cisco Modeling Labs server has started the simulation launch, a confirmation dialog box appears with an identifier assigned to the simulation by the Cisco Modeling Labs server.

When the nodes are running on the Cisco Modeling Labs server, they are displayed in the **Simulations** view with the status as [ACTIVE], as shown in the following figure.

Figure 2: Simulation Launched in Simulation View
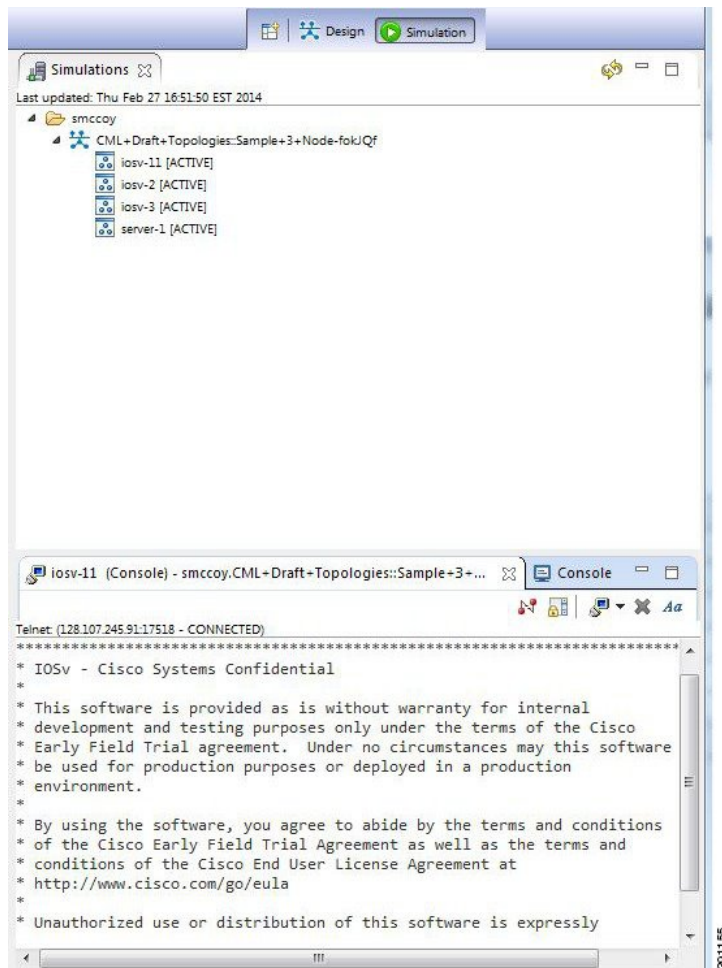
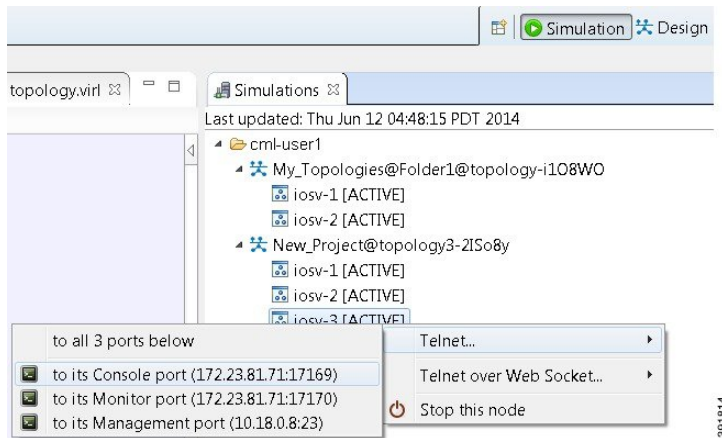# Connect to a Simulation Node Console

Before You Begin

- Launch a simulation.

- Ensure that the **Simulation** perspective is active.

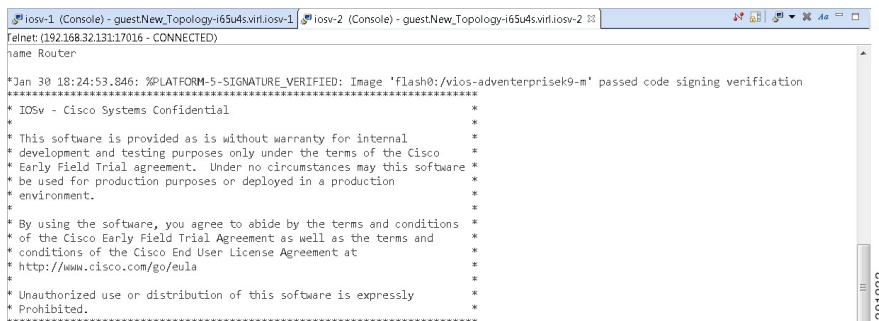• Ensure that the **Simulations** view and **Console** view are displayed.

Step 1   To connect to a console for a specific node, right-click the node in the **Simulations** view and choose **Telnet** > **to its Console port**.

Figure 3: Connecting to a Node Console

A new **Terminal** view opens.

Figure 4: Terminal View

Step 2   To disconnect a terminal from the simulation, click **Disconnect** in the **Terminal** view toolbar or click the **Close** icon in the **Terminal** view.

Note   When you disconnect or close a terminal view, all text in the view is discarded.

Tip   After you disconnect a terminal, but do not close the terminal window, you can press **Enter** to reconnect the terminal.

Stop an Entire Simulation

There are several methods for stopping an entire simulation. These are discussed in the following sections.

Method 1: Stop an Entire Simulation from the Toolbar

Step 1 In the toolbar, click the **Stop Simulations** button.
A **Stop Simulation(s)** dialog box appears.

Step 2 In the **Simulation** column, click once to highlight the simulation to stop.

Step 3 (Optional) To save the configurations, click the adjacent setting in the **Extract Configurations?** column until the prompt changes to **Yes**.
Note Configurations for server nodes are not extracted.

Step 4 (Optional) To close the internal terminals associated with the simulation, click the adjacent setting in the **Close Terminals?** column until the prompt changes to **Yes**.
Note External terminal connections are not stopped as part of this operation and must be closed manually.

Step 5 To stop the simulation, click the adjacent setting in the **Stop Simulation?** column until the prompt changes to **Yes**.

Step 6 Click **OK** to confirm stopping the simulation, or click **Cancel** to leave the simulation running.

Figure 5: Stop Simulation(s) Dialog Box
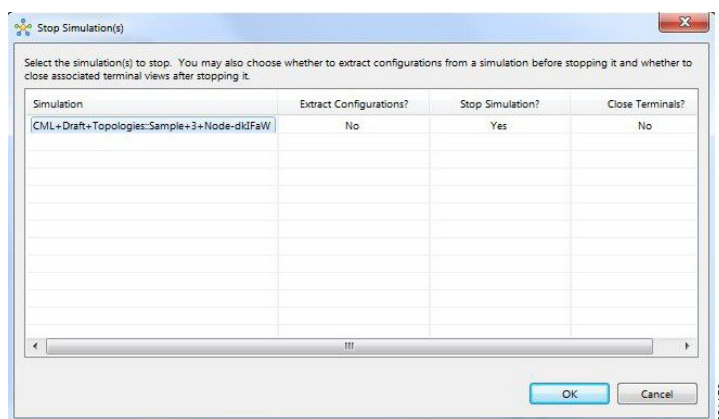
On Mac OS X, you update the values for **Extract Configurations?**, **Stop Simulation?** and **Close Terminals?** in the columns directly. You do not need to select the name of the simulation.

Note When stopping a simulation, you cannot attempt to stop individual nodes running in the simulation at the same time. This behavior is not supported.

Method 2: Stop an Entire Simulation from the Simulations View

Step 1 In the **Simulations** view, right-click the **Simulation** name and select **Stop Simulation**. The **Stop simulation confirmation** dialog box appears.

> • (Optional) Check the **Extract configurations before stopping simulation** check box to save the current configurations.
>
> • (Optional) Check the **Close associated terminal views** check box to close all the open internal terminals associated with the specific simulation. External terminals are not closed.
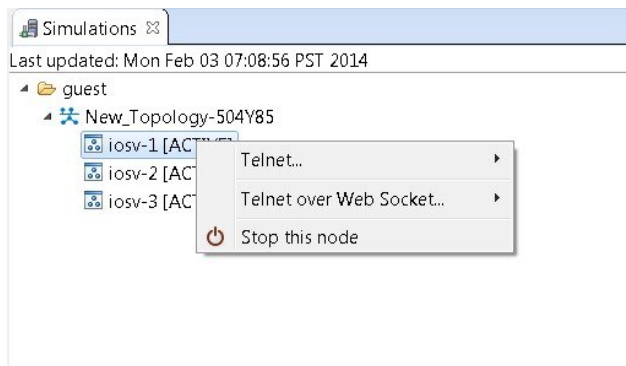
Step 2 Click **OK** to stop the simulation.
Once selected, all nodes in the simulation start shutting down. It may take a few minutes for the simulation to shut down completely and to disappear from the **Simulations** view.

> **Note** When stopping a simulation, you cannot attempt to stop individual nodes running in the simulation at the same time. This behavior is not supported. Note too for instances where a user account expires, any running simulations for that user continue to run. Since the user account can no longer log into stop them, they will remain active until the next system reboot or until the system administrator explicitly stops them.

Stop a Single Node

Step 1 In the **Simulations** view, right-click the node to stop and select **Stop this node**.

The **Are you sure** dialog box appears.

Step 2 Click **OK** to stop the simulation. Alternatively, click **Cancel** to abandon the operation and return to the simulation.

> **Note** When you click **OK**, the node stops without saving any changes to the configuration.
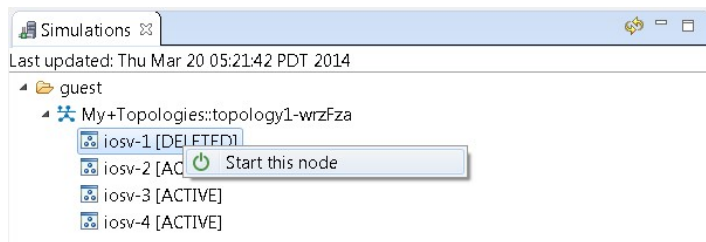
When a node is stopped, its status changes to [DELETED].

What to Do Next

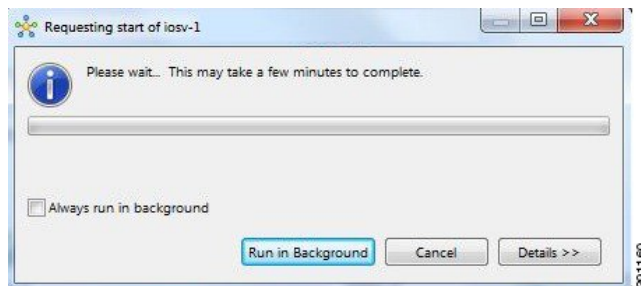- (Optional) Start a node.

- (Optional) Stop the simulation.

Start a Single Node

Step 1　Right-click a stopped node.
When a node is stopped, its status changes to [DELETED].

Step 2　Click **Start this node**.

The **Requesting start** dialog box appears.

Step 3　Choose one or more of the following actions:

a) Check the **Always run in the background** check box. All future node start requests, stop requests, and simulation launch requests run in the background and do not display dialog boxes.
   Note　To control the background setting, choose **File** > **Preferences** > **General**.

b) Click **Run in background**.
   The dialog box closes while the node simulation stops.

c) Click **Cancel** to return to the **Simulations** view.

d) Take no action and the node simulation restarts momentarily.

Tip   When you click **Run in background**, the status bar displays a progress icon. Click the icon to display a compact view of the progress. If an error is encountered, the icon shows a red **X**. Click the error icon to display the error dialog box.
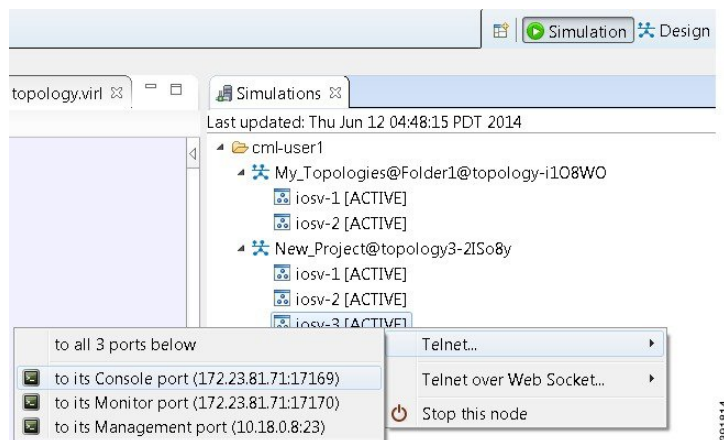
What to Do Next

- (Optional) Stop the simulation.

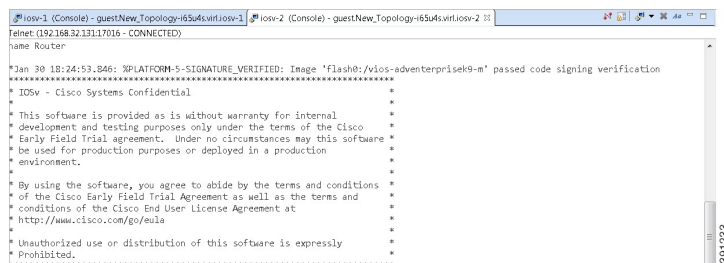# Modify a Node Configuration in the Simulation

Step 1   Connect to a console on a router node. Right-click the node in the **Simulations** view and choose **Telnet > to its Console port**.

Figure 6: Connecting to a Node Console

A new **Terminal** view opens.

Figure 7: Terminal View

Step 2   If no banner or router prompt is visible, press **Enter**.

You are now working with the operating system running on the node, for example, Cisco IOS software.

Step 3 Use the operating system commands to view or modify the node configuration.

Note Changes you make to the configuration do not appear in the canvas of the **Topology Editor**.

Extract and Save Modified Configurations

Before You Begin

- You have modified a configuration within one or more nodes running within the simulation and want to save the changes.

- Ensure that all routers in the simulation are operational before attempting to extract their configurations.

Step 1 In the **Simulations** view, right-click the topology name, making sure not to click the node name, and select **Extract Configurations**.
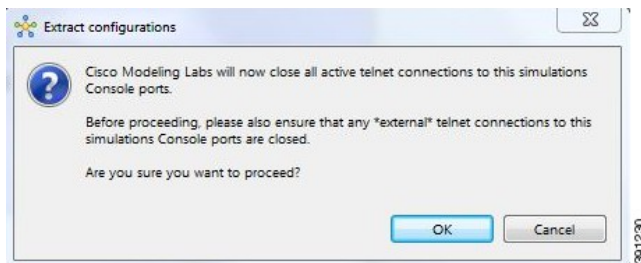
Figure 8: Simulations View Context Menu
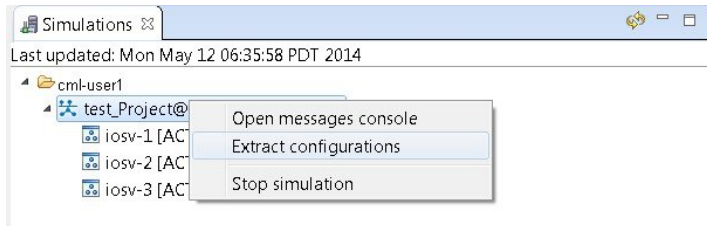
A confirmation dialog box appears.

Step 2 Ensure that all external Telnet connections to the simulation are closed before proceeding.

Note You must close all external Telnet connections to the simulation before you can proceed.

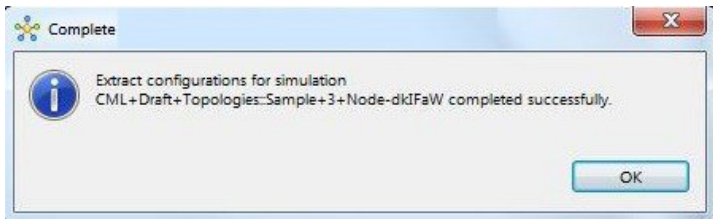*Figure 9: Extract Configurations Dialog Box*

Step 3 Click **OK**.

A dialog box appears.

Step 4 Click **Run in Backgroud**.
When the extraction is complete, a message is displayed.

Figure 10: Extraction Complete Message

Note The configuration information is extracted and saved in the *filename.vir*l file that corresponds to the topology. For example, the file New_Topology that is running as a New_Topology-dkIFaW simulation has its configuration extracted to the file **New_Topology.virl**.

Step 5 Click **OK**.

What to Do Next

- (Optional) Stop the simulation.
- (Optional) Stop a single node in the simulation.