



Simulate the Topology

- [Simulate the Topology Overview, page 1](#)
- [Launch a Simulation, page 2](#)
- [Connect to a Simulation Node Console, page 18](#)
- [Start a Single Node, page 24](#)
- [Stop a Simulation, page 30](#)
- [Stop a Single Node, page 34](#)
- [Modify a Node Configuration in the Simulation, page 37](#)
- [Extract and Save Modified Configurations, page 42](#)
- [Linux Server Snapshot Support, page 45](#)

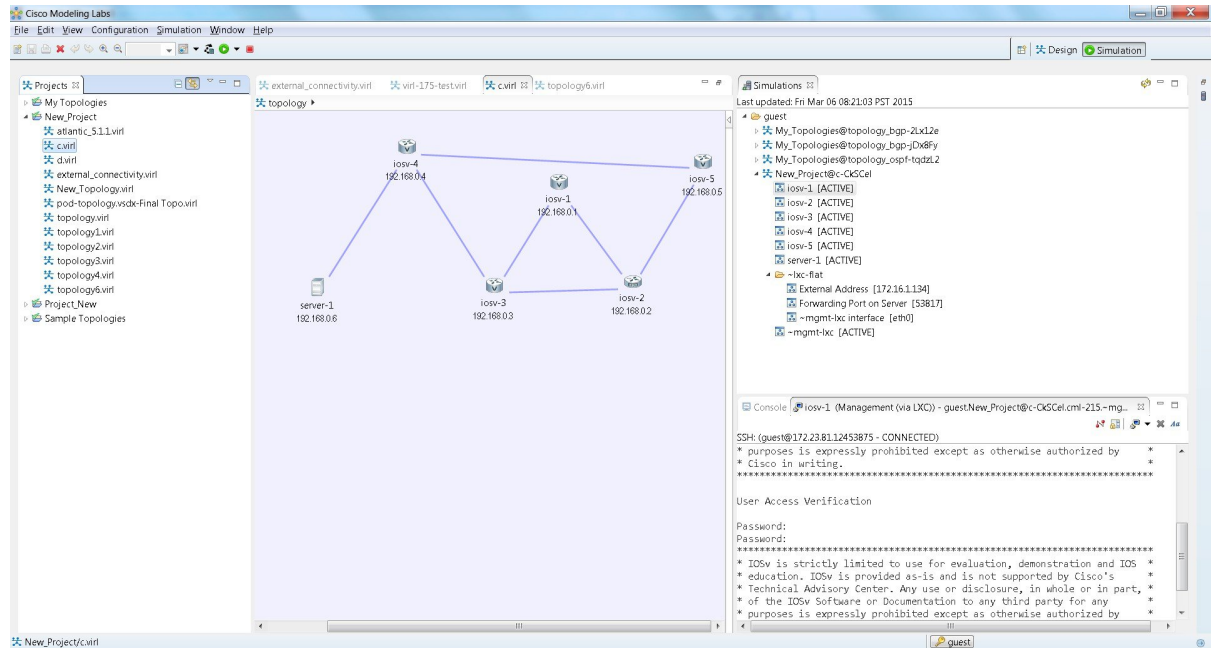
Simulate the Topology Overview

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 **Simulations** 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 [Customize 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, the number of nodes you can run is limited by the resources available on the server and the number of licenses available. Currently, the design limit is 1,000 nodes.

Launch a Simulation

To launch a simulation, complete the following steps.

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 autogeneration, 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.

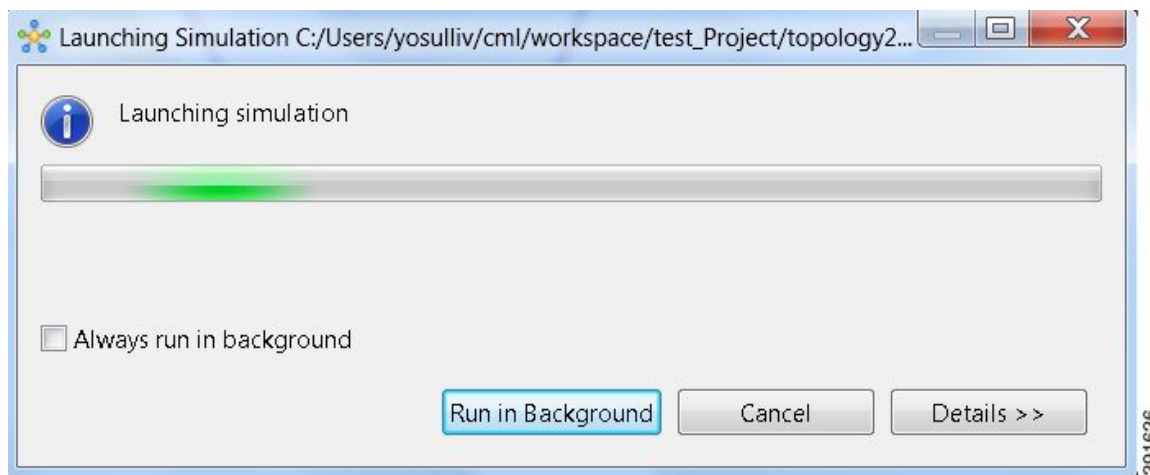
Step 1

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:



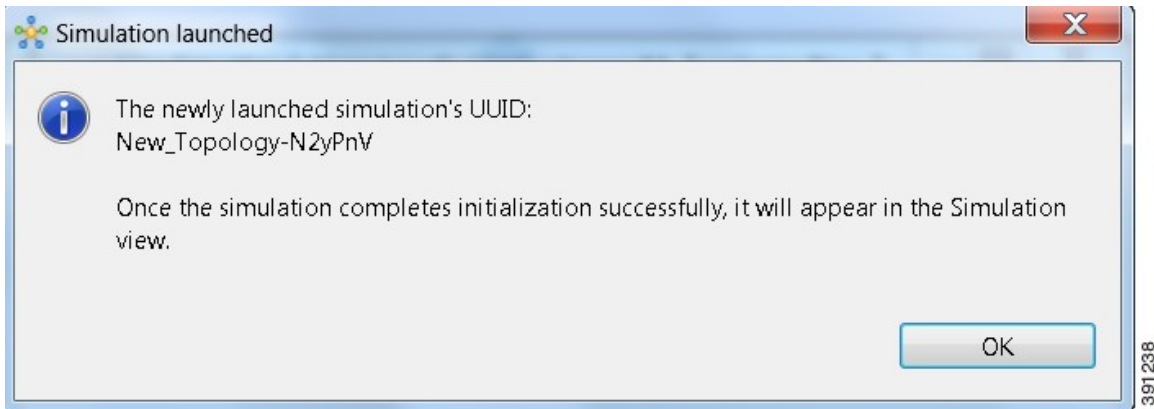
- Check the **Always run in the background** check box. All future node start requests, stop requests, and simulation launch requests run in the background and do not display dialog boxes.

Note To control the background setting, choose **File > Preferences > General**.

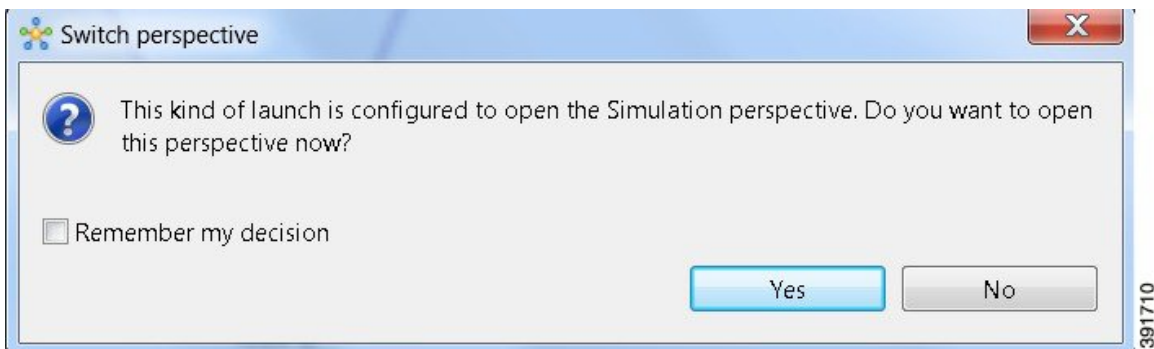
- Click **Run in Background**. The dialog box closes when the node simulation starts.
- Click **Cancel** to return to the **Design** view.
- Take no action, and the node simulation launches momentarily.

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**.

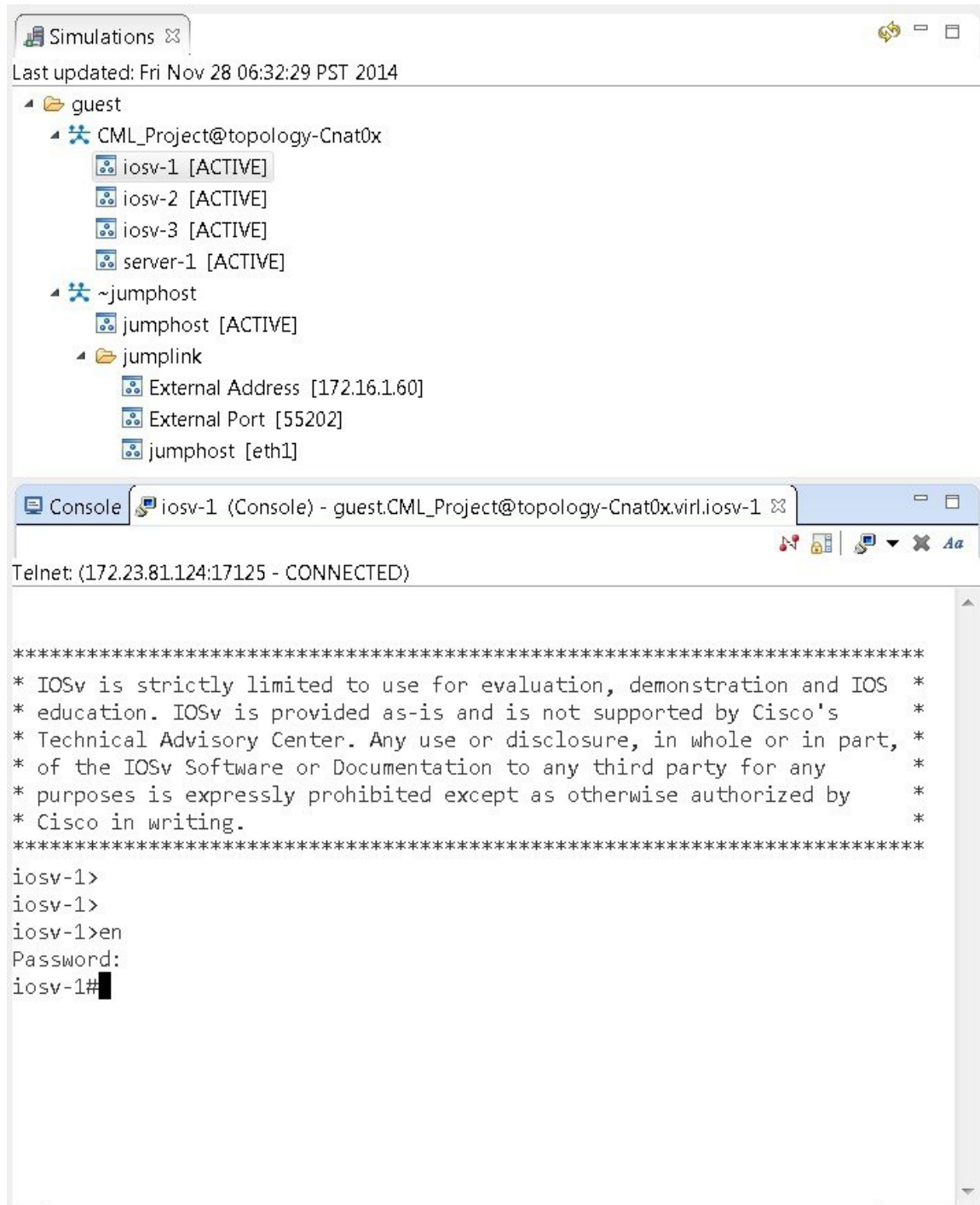


Note We recommend that you switch to the **Simulation** perspective to view the running simulation and to connect to node consoles.

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 **[ACTIVE]**, as shown in the following figure.

Figure 2: Simulation Launched in Simulations View



Jumphost Virtual Machine (VM)

The jumphost VM is the default method for accessing the management network of a running simulation. The jumphost node runs in a separate simulation named `~jumphost` and has two interfaces, `eth0` in the project/user management network and `eth1` in the FLAT network.

The purpose of the jumphost is to provide an access point into a simulated network that remains fixed, in that there is a single external IP address or port that the user can access. A user can access the jumphost and then access all the nodes inside the simulation.

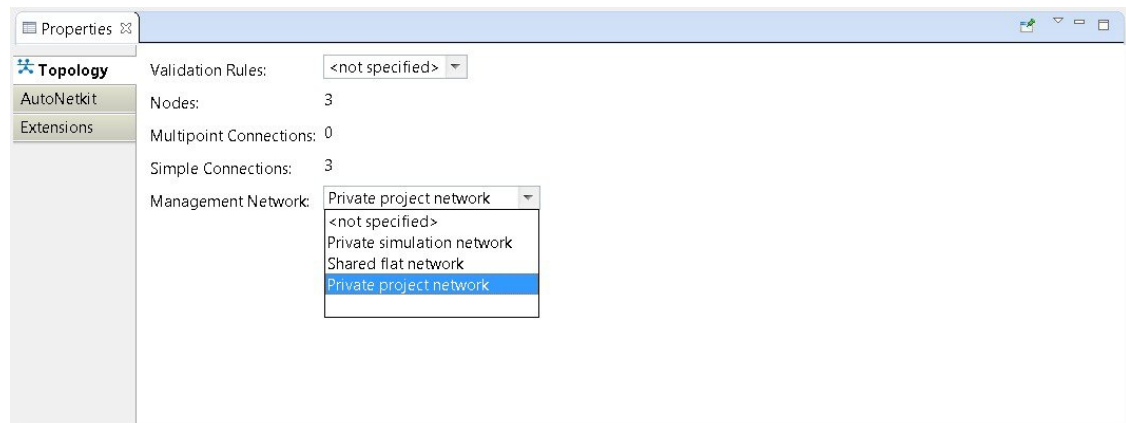
Cisco Modeling Labs provides two implementations of jumphost:

- A VM: Based on the server VM image type.
- A Linux container: A lighter weight form of a jumphost. See [Linux Container \(LXC\)](#), on page 7 for more information.

The VM implementation is costly in terms of the memory and CPU used when a jumphost is instantiated. However, since it is a full-blown server VM, there is value to it, in that you can install and run any application on it.

To select a jumphost VM, in the Cisco Modeling Labs client, choose **Properties > Topology > Management Network > Private project network** or **Shared flat network**.

Figure 3: Setting the Jumphost Option



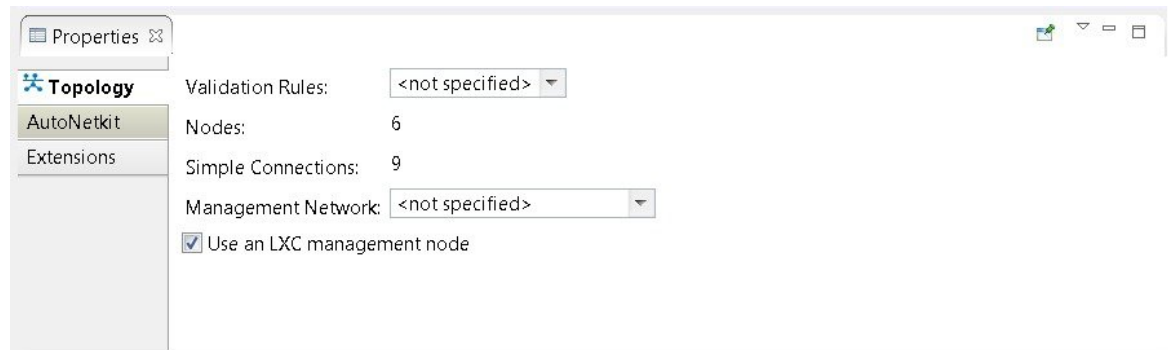
The jumphost VM is per user; a user can create multiple simulations, but only one corresponding jumphost is created. Therefore, depending on the type of simulation you are running, you can choose between the two implementations.

Linux Container (LXC)

An LXC provides a means of accessing a topology rather than having to create a full Linux server VM. All nodes in the network are connected to a hidden OOB management network that uses the first interface on each of the nodes.

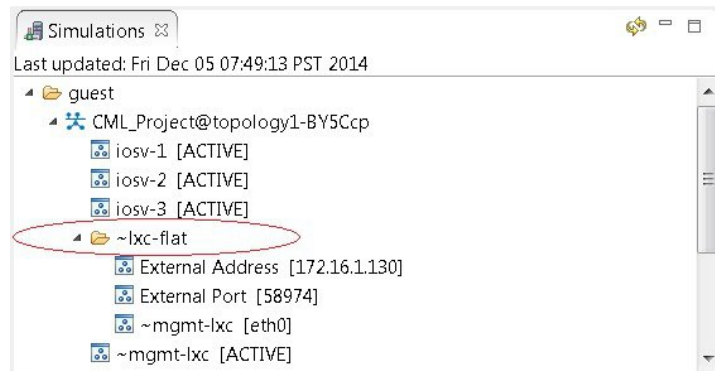
In the Cisco Modeling Labs client, it is enabled by selecting the management network type **Private simulation network** for your topology. The LXC is automatically connected into this hidden OOB management network to which all VMs in your simulation are connected. This enables you to connect into each VM via its management Ethernet port, removing the need to use the console port connection method.

Figure 4: Setting the LXC Option



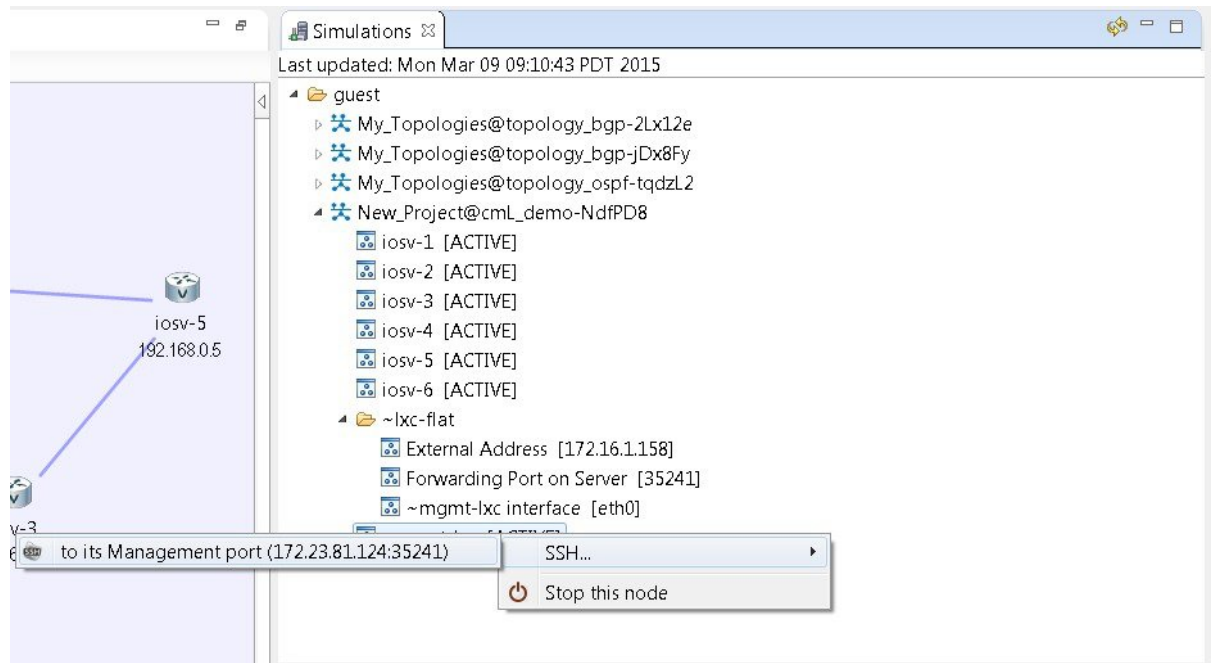
The LXC is operational when your simulation is active and terminates when your simulation stops. LXC uses a significantly smaller memory footprint than the Linux server VM. The LXC automatically gets an IP address on the FLAT network.

Figure 5: LXC on the FLAT Network



LXC facilitates SSH access to the VMs; it provides direct SSH access to each VM running inside the simulation. Telnet is not supported. As shown below, LXC is accessed by right-clicking the `~mgmt_lxc [ACTIVE]` node and selecting `SSH` from the list.

Figure 6: Accessing LXC



The LXC is automatically spun up and provides a jump host point for access into the network. Connecting to the LXC means you can see the interfaces to the outside world and to the OOB network inside your simulation.

A connection is opened from the Cisco Modeling Labs client to a port on the Cisco Modeling Labs server and is forwarded to the LXC. The LXC, in turn, opens a connection to the Management Interface (Gi0/0) of the VM inside the simulation to the node instance.

Static Port Assignment to the LXC

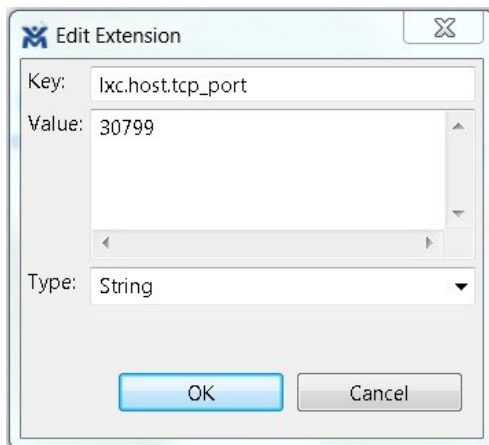
When the management network property **Private simulation network** is set, Cisco Modeling Labs assigns a random port for SSH port access to the LXC. However, you can statically define this by setting an extension on your topology.

To set an extension for your topology, complete the following steps.

-
- Step 1** Click on the canvas to open the **Topology** tab in the **Properties** view.
 - Step 2** Click the **Extensions** tab.
 - Step 3** Click the **Add new extension** icon.
The **Edit Extension** dialog box appears.
 - Step 4** Enter the following values:

- **Key:** `lxc.host.tcp_port`
- **Value:** `30799`
- **Type:** `String` (from the drop-down list)

Figure 7: Add a New Extension



- Step 5** Click **OK** to add the new extension.
When the LXC starts, it will be bound to the TCP port specified in the new extension.
-

Launch a Phased Simulation

On occasions, you may need to start your simulation in phases rather than having all nodes launched at the same time. This functionality is facilitated by the **Exclude node from simulation launch** check box, which allows you to pick and choose which nodes to start.

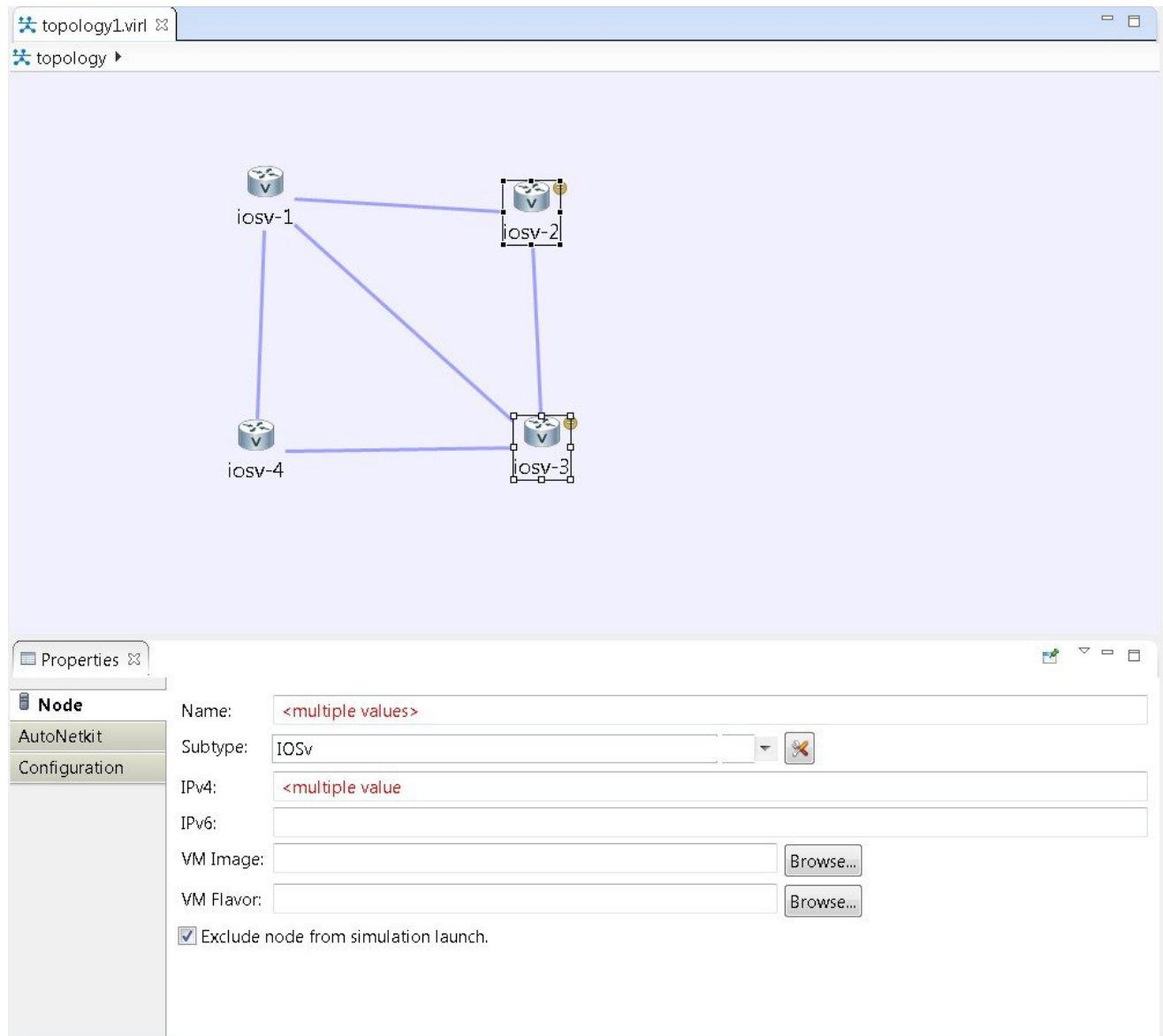
To launch a phased simulation, complete the following steps.

- Step 1** With the applicable topology open, click on the canvas and move the selection area over the nodes to be excluded from the running simulation. (Alternatively, you can double-click a specific node, hold down the **Shift** key, and select the remaining nodes.)

The **Properties > Node** view opens.

Step 2 In the **Properties > Node** view, check the **Exclude node from simulation launch** check box.

Figure 8: Nodes Excluded from Simulation Start



Step 3 Save your topology using **Ctrl-S**.

Note You can set this property for each individual node, if you prefer. Select the node on the canvas, and select the **Exclude node from simulation launch** check box in the **Properties > Node** view.

Step 4 From the toolbar, click the **Build Initial Configurations** button to build the node configurations.

Step 5 From the toolbar, click the **Launch Simulation** button.
The simulation launches.

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

Note In the **Console** view, you can see the message **Node '<node name>' is excluded from automatic start** for the excluded nodes, and in the **Simulations** view, you can see that the excluded nodes have the state **[ABSENT]**.

Figure 9: Phased Simulation Launched

Simulations

Last updated: Sat Aug 15 12:37:18 PDT 2015

- guest
 - New_Project@topology1-RGXfvd
 - iosv-1 [ACTIVE]
 - iosv-2 [ABSENT]
 - iosv-3 [ABSENT]
 - iosv-4 [ACTIVE]
 - ~lxc-flat
 - External Address [172.16.1.75]
 - Forwarding Port on Server [45069]
 - ~mgmt-lxc interface [eth0]
 - ~mgmt-lxc [ACTIVE]

Console

```

topology1.virl simulation New_Project@topology1-RGXfvd
(INFO) [Aug/15/2015 19:36:41] Node "iosv-2" is excluded from automatic start
(INFO) [Aug/15/2015 19:36:42] Node "iosv-3" is excluded from automatic start
(INFO) [Aug/15/2015 19:36:42] Session "New_Project@topology1-RGXfvd": nodes can start
(INFO) [Aug/15/2015 19:36:45] Starting node "iosv-4"
(INFO) [Aug/15/2015 19:36:45] Starting node "iosv-1"
(INFO) [Aug/15/2015 19:36:51] Starting node "~mgmt-lxc"
(INFO) [Aug/15/2015 19:36:59] Node "iosv-1" state changed from ABSENT to BUILDING
(INFO) [Aug/15/2015 19:36:59] Node "iosv-4" state changed from ABSENT to BUILDING
(INFO) [Aug/15/2015 19:37:16] Node "iosv-1" state changed from BUILDING to ACTIVE
(INFO) [Aug/15/2015 19:37:16] Node "iosv-4" state changed from BUILDING to ACTIVE
  
```

Launch a Time-Limited Simulation

In circumstances where you need to run a simulation for a specified time, complete the following steps.

Step 1 From the toolbar, click the down arrow to the right of the **Launch Simulation** button.

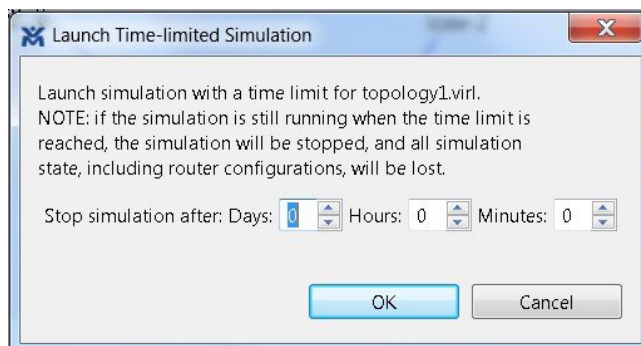
Step 2 Click **Launch Simulation (with time limit)**.

Figure 10: Launch Simulation with Time Limit Option



The **Launch Time-limited Simulation** dialog box is displayed.

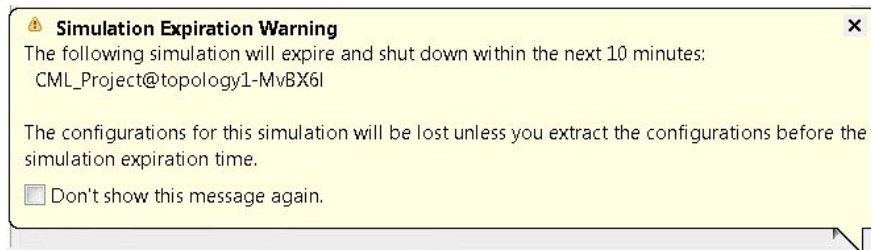
Figure 11: Launch a Time-Limited Simulation



Step 3 Set a time duration for the simulation by entering details for **Days**, **Hours**, and **Minutes** by using either the up and down arrows or entering the values directly. Click **OK** to apply your time limit to the simulation. The simulation launches.

Note An expiration warning is displayed ten minutes (or less) before your simulation is due to expire.

Figure 12: Simulation Expiration Warning



Note We recommend that you extract configurations for your simulation before it expires. You can reset the time limit for a running simulation in the **User Workspace Management** interface. See [Reset the Time Limit on a Running Simulation](#), on page 14 for more information.

Reset the Time Limit on a Running Simulation

You can extend or reduce the time limit set for a running simulation in the **User Workspace Management** interface. To do this, complete the following steps.

- Step 1** Log in to the **User Workspace Management** interface. On the **Overview** page, information on running simulations is displayed. Move to the applicable simulation under the **Session** heading. If your simulation is due to expire in ten minutes or less, the simulation name is displayed in red.
- Step 2** Under the **Options** column, click the down arrow and click **Set expiration**.

Figure 13: Soon to Expire Simulation

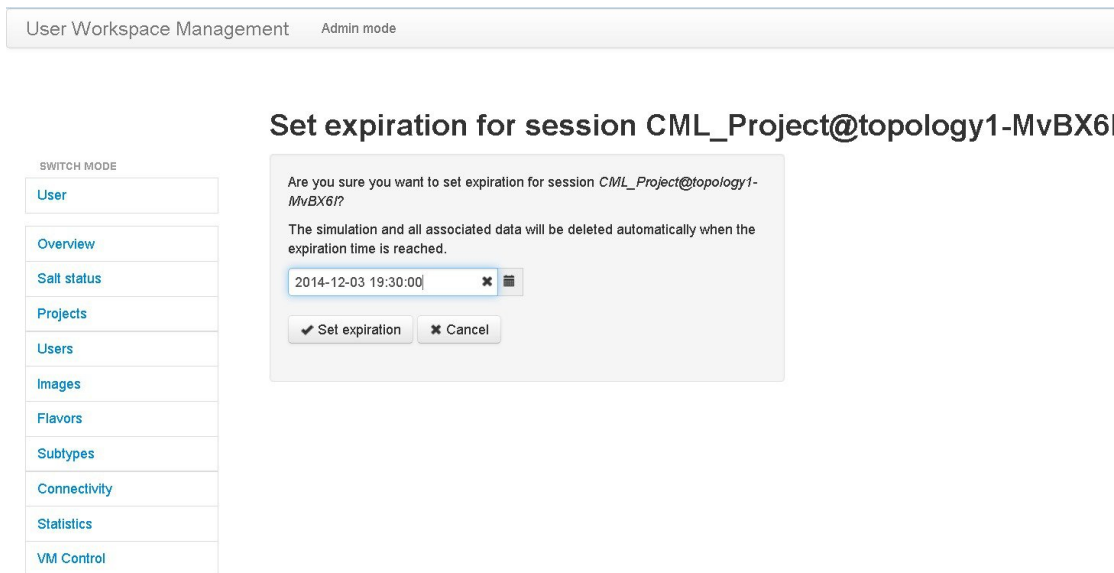
Sessions Request to stop all

Sessions of project *guest* Request to stop selected

Session	User	Project	Status	Expires	Options
CML_Project@topology1-MvBX6l	guest	guest	ACTIVE	2014-12-03 19:18:00 (in 0d 00h 09m)	<div style="border: 1px solid #ccc; padding: 2px;"> Request stop Set expiration </div>
~jumphost	guest	guest	ACTIVE	never	

Step 3 The **Set expiration for session** page is displayed. In the date and time field, enter either a new expiration date and time, a date only, or a time only for the simulation.

Figure 14: Set Expiration for Session Page



Step 4 Click **Set expiration** to save the changes. The time limit for the simulation is updated.

Control Interface States

In a running simulation, you are able to change the state of the network interface by bringing it up or down.



Note This changes the state of the underlying communication infrastructure, not the interface state of the virtual machine.

To control the state of an interface, complete the following steps.

- Step 1** Log in to the **User Workspace Management** interface.
- Note** You must log in as a user other than the uwadmin user, for example, guest.
- Step 2** On the **Overview** page, under **Sessions**, choose the applicable running session.

A list of active virtual machines and interfaces is displayed.

Step 3 Scroll down to the **Interfaces** section and choose the applicable virtual machine.

Step 4 From the applicable **Options** drop-down list, click **Update admin state**.

Figure 15: Interface State Control Option

Interfaces

Node	Interface name	Interface up	Network subtype	Network name	IP Addresses	Options
iosv-1	GigabitEthernet0/0	True	SESSION MGMT	mgmt	10.255.0.5 / 16	▼
iosv-1	GigabitEthernet0/1	True	SIMPLE	iosv-1-to-iosv-3	unassigned	▼
iosv-1	GigabitEthernet0/2	True	SIMPLE	iosv-1-to-iosv-2	unassigned	Update admin state
iosv-1	GigabitEthernet0/3	True	SIMPLE	iosv-1-to-iosv-4	unassigned	▼
iosv-1	GigabitEthernet0/4	True	SIMPLE	iosv-1-to-iosv-6	unassigned	▼
iosv-2	GigabitEthernet0/0	True	SESSION MGMT	mgmt	10.255.0.6 / 16	▼
iosv-2	GigabitEthernet0/1	True	SIMPLE	iosv-1-to-iosv-2	unassigned	▼
iosv-2	GigabitEthernet0/2	True	SIMPLE	iosv-2-to-iosv-3	unassigned	▼
iosv-2	GigabitEthernet0/3	True	SIMPLE	iosv-2-to-iosv-4	unassigned	▼
iosv-2	GigabitEthernet0/4	True	SIMPLE	iosv-2-to-iosv-5	unassigned	▼
iosv-3	GigabitEthernet0/0	True	SESSION MGMT	mgmt	10.255.0.7 / 16	▼

Depending on the current state of the interface, either a **Bring Down** or **Bring Up** page is displayed. In this case, the **Bring Down** page is displayed.

Figure 16: Bring Down the Applicable Interface

Bring down interface GigabitEthernet0/1 on node iosv-1

Are you sure you want to bring down interface **GigabitEthernet0/1** on node **iosv-1**?

Step 5 Click **Bring down** to bring down the network interface.

A message is displayed indicating that the interface has been brought down.

Figure 17: Interface Successfully Brought Down



Session Sample_Topologies@6-node-triangle-WPUM0y details

Nodes

Node	Subtype	State	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
iosv-3	IOSv	ACTIVE	▼
iosv-4	IOSv	ACTIVE	▼
iosv-5	IOSv	ACTIVE	▼
iosv-6	IOSv	ACTIVE	▼
~mgmt-lxc	mgmt-lxc	ACTIVE	

406558

Step 6 To bring up the interface at a later stage, click **Update admin state** again. The **Bring up** page displayed.

Figure 18: Bring Up the Applicable Interface

Bring up interface GigabitEthernet0/1 on node iosv-1

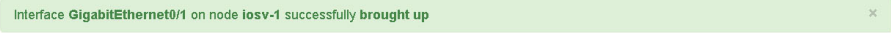


406559

Step 7 Click **Bring up** to bring up the network interface.

A message is displayed indicating that the interface has been brought up.

Figure 19: Interface Successfully Brought Up



Interface GigabitEthernet0/1 on node iosv-1 successfully brought up

Session Sample_Topologies@6-node-triangle-WPUM0y details

Nodes

Node	Subtype	State	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
iosv-3	IOSv	ACTIVE	▼
iosv-4	IOSv	ACTIVE	▼
iosv-5	IOSv	ACTIVE	▼
iosv-6	IOSv	ACTIVE	▼
~mgmt-kc	mgmt-kc	ACTIVE	

406560

Connect to a Simulation Node Console

Cisco Modeling Labs provides the capability for you to connect to your nodes via SSH and Telnet. You can start an SSH session, which connects into the node via the LXC, as described in [Linux Container \(LXC\)](#), on page 7.

This access method makes use of SSH to the LXC and then Telnet from the LXC to the nodes running inside the simulation. This does not use the console port of the nodes and is more reliable and faster to use.

Connect to a Simulation Node Console via SSH

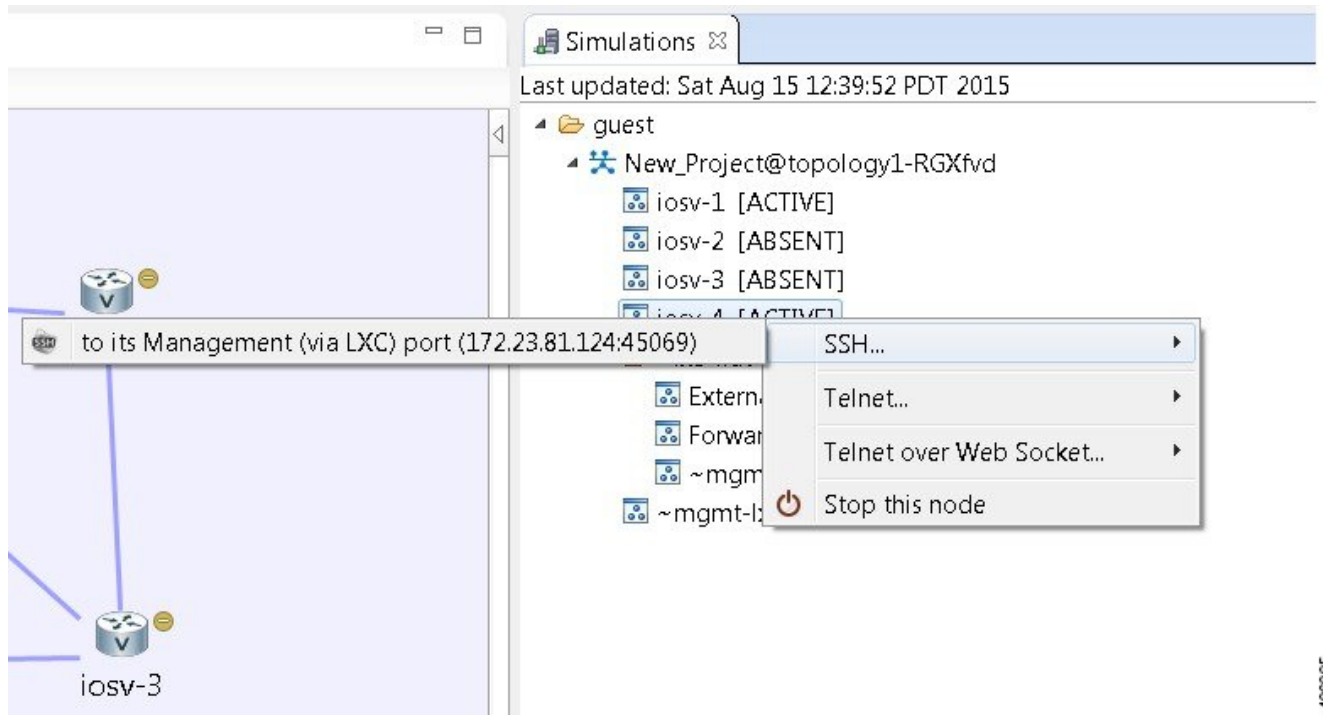
To connect to a simulation node console, complete the following steps.

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 **SSH > to its Management (via LXC) port**.

Figure 20: Connecting to a Node Console via SSH



406095

A new **Terminal** view opens.

Figure 21: Terminal View

```

Console | iosv-4 (Management (via LXC)) - guest.New_Project@topology1-RGXfvd.virt~mgmt-lxc
SSH: (guest@172.23.81.12445127 - CONNECTED)
guest@mgmt-RGXfvd$
guest@mgmt-RGXfvd$
guest@mgmt-RGXfvd$ telnet 10.255.0.110
Trying 10.255.0.110...
Connected to 10.255.0.110.
Escape character is '^]'.

*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing. *
*****

User Access Verification

Password:
*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing. *
*****

iosv-4>

```

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.

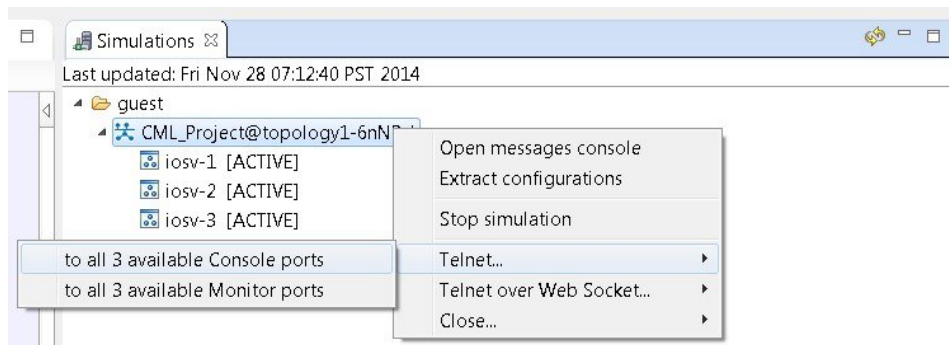
Tip When you disconnect a terminal but do not close the **Terminal** window, you can press **Enter** to reconnect the terminal.

Connect to Multiple Simulation Node Consoles

To connect to all consoles for all nodes in a running simulation, complete the following steps.

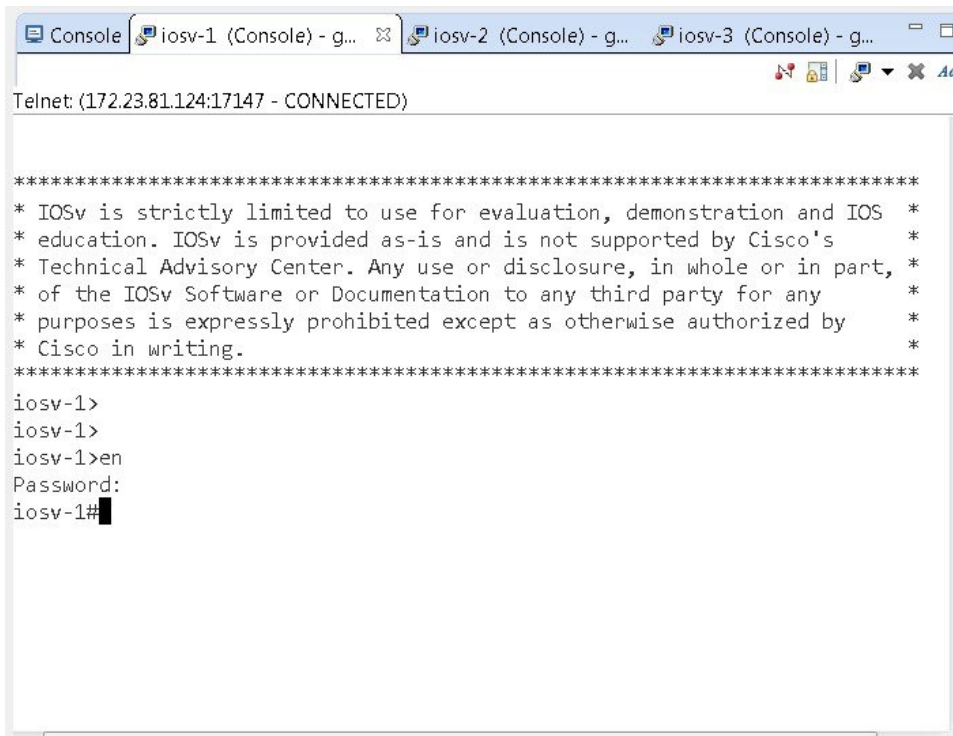
Step 1 Right-click the simulation in the **Simulations** view and choose **Telnet > to all <number> available Console ports**.

Figure 22: Connect to Multiple Node Consoles



A new **Terminal** view opens for all console ports.

Figure 23: 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 When you disconnect a terminal but do not close the **Terminal** window, you can press **Enter** to reconnect the terminal.

Terminal Multiplexer Functionality

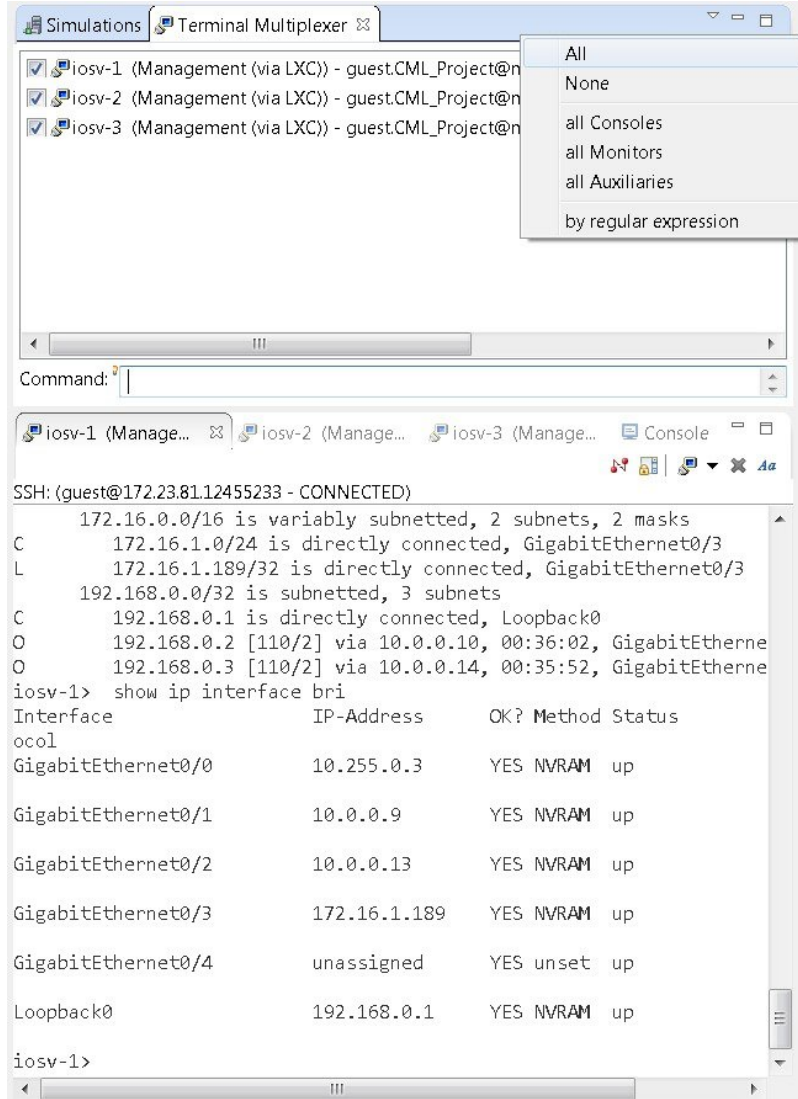
A terminal multiplexer is available for use with the Cisco Modeling Labs client. It permits a number of terminals to be accessed and controlled from a single terminal. Terminals can be detached to run in the background and then reattached later.

The terminal multiplexer is available from **Window > Show View > Other > General > Terminal Multiplexer**.

When the terminal multiplexer starts up, a status line at the bottom of the terminal displays information on the current session and is used to enter interactive commands. It lists all of the active console sessions. You

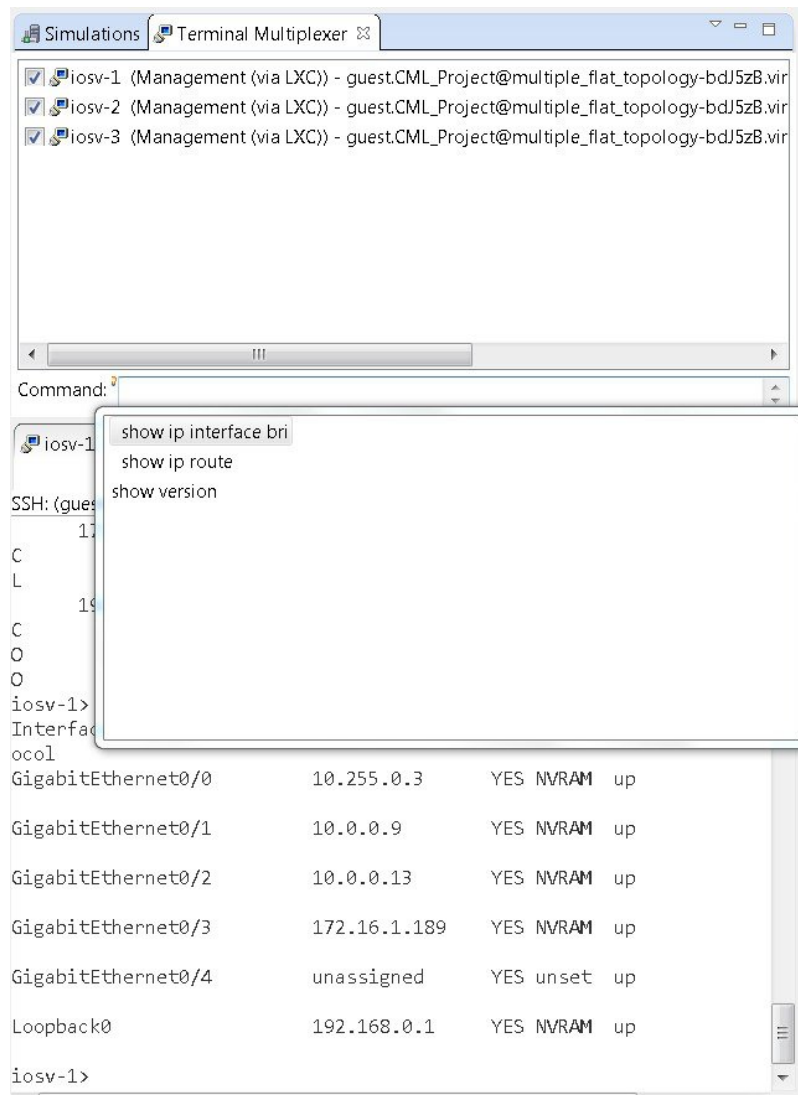
can select console sessions individually or all together from the **View Menu** drop-down list. Keyboard commands entered in the command-line text box are broadcast to the selected sessions.

Figure 24: Terminal Multiplexer



The terminal multiplexer also provides a command-line history, which you can access using **Ctrl-Space**.

Figure 25: Accessing the Command-Line History



Start a Single Node

To start a single node, complete the following steps.

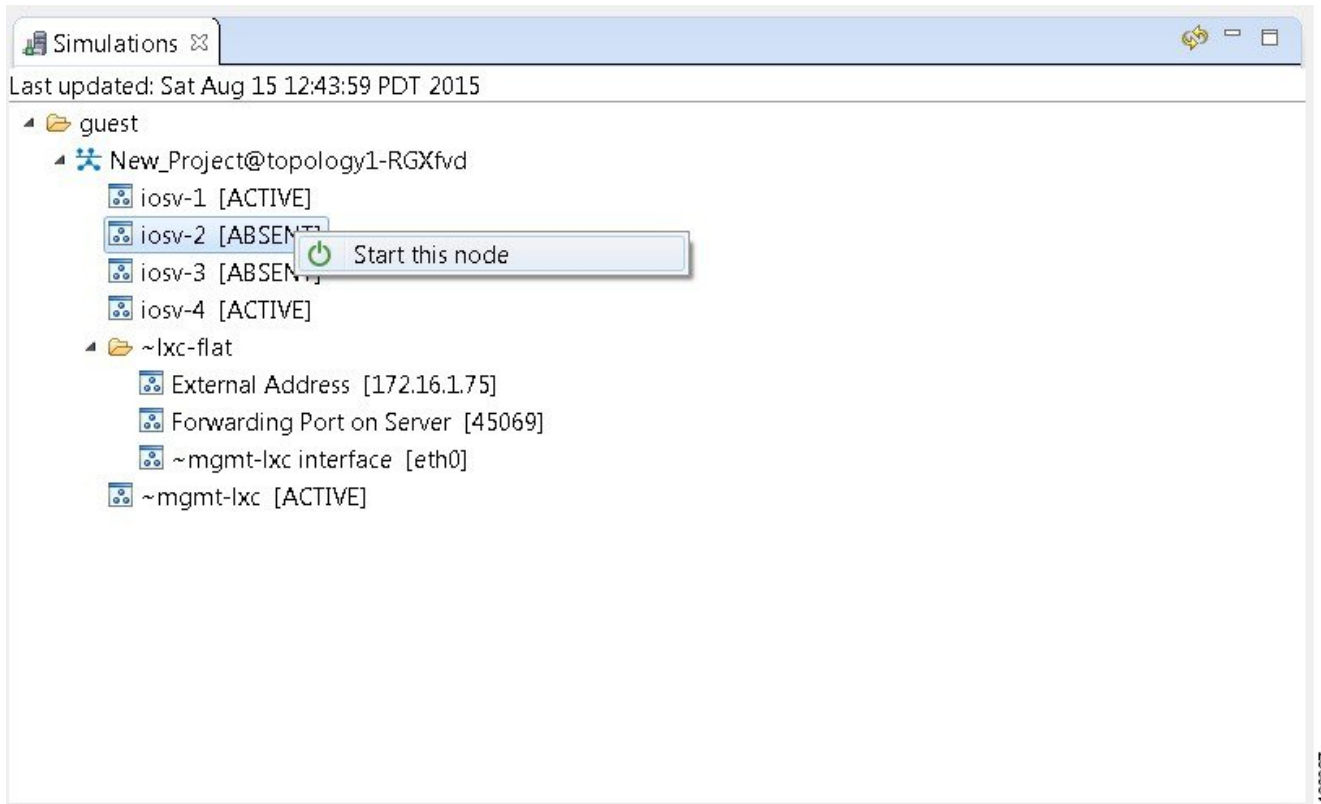
Step 1 Right-click a stopped node.

Step 2

When a node is stopped, its status is shown as [ABSENT].

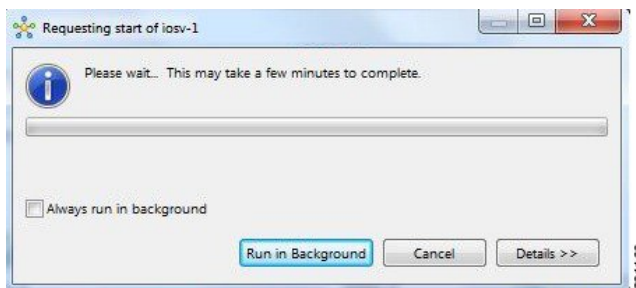
Click **Start this node**.

Figure 26: Start a Single Node



The **Requesting start** dialog box appears.

Figure 27: Requesting Start Dialog Box



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 displays a red **X**. Click the error icon to display the error dialog box.

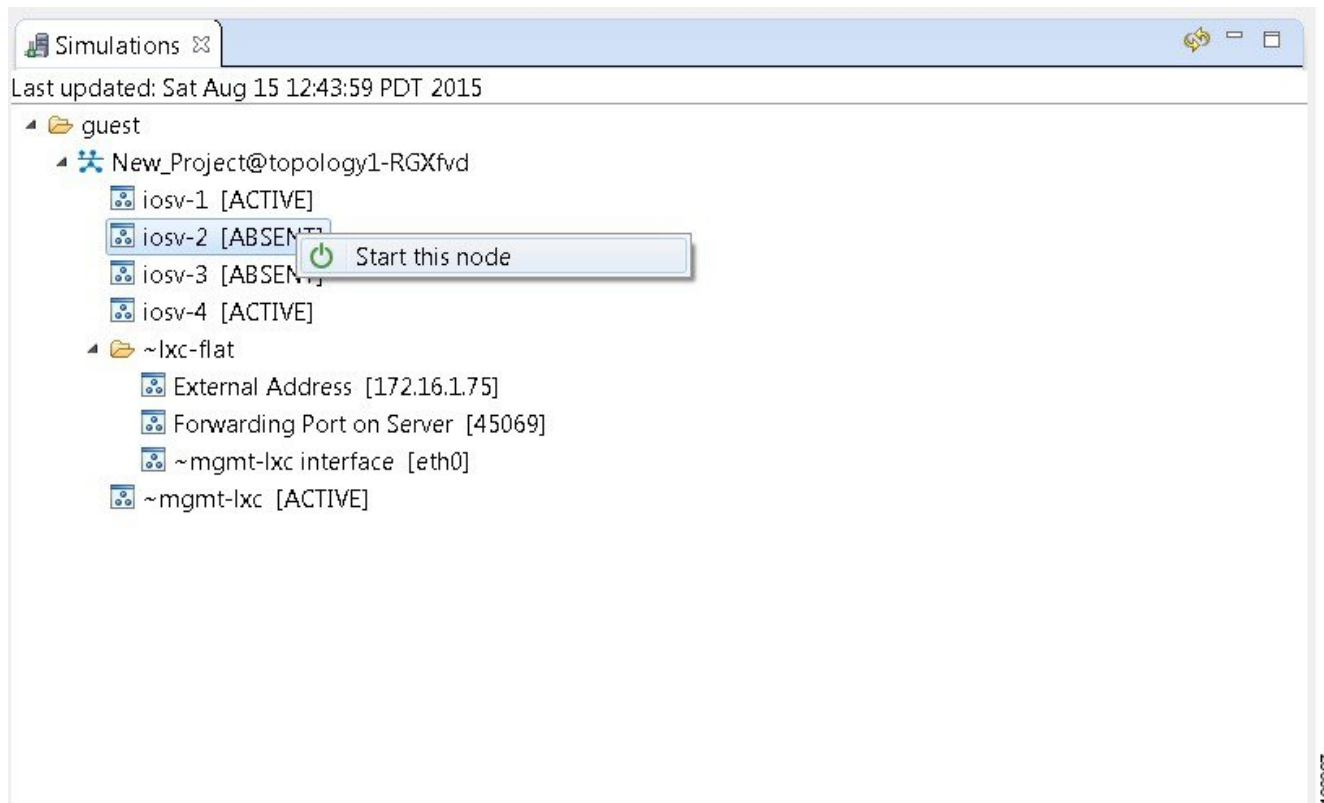
Start a Node in a Running Simulation

In cases where a phased simulation is running, you can later start those nodes not started with the initial simulation. To start a node in a running simulation, complete the following steps.

-
- Step 1** In the **Simulations** view, right-click the node.
- Note** A node not yet started has the status **[ABSENT]**.

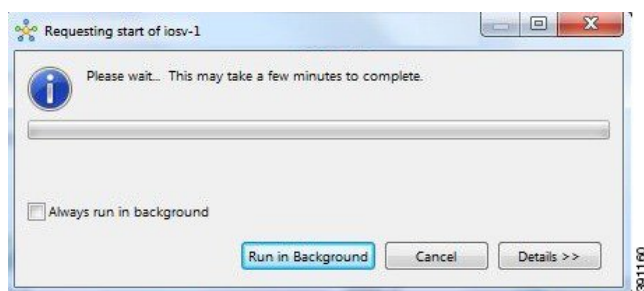
Step 2 Click **Start this node**.

Figure 28: Start This Node Option



The **Requesting start** dialog box appears.

Figure 29: Requesting Start Dialog Box



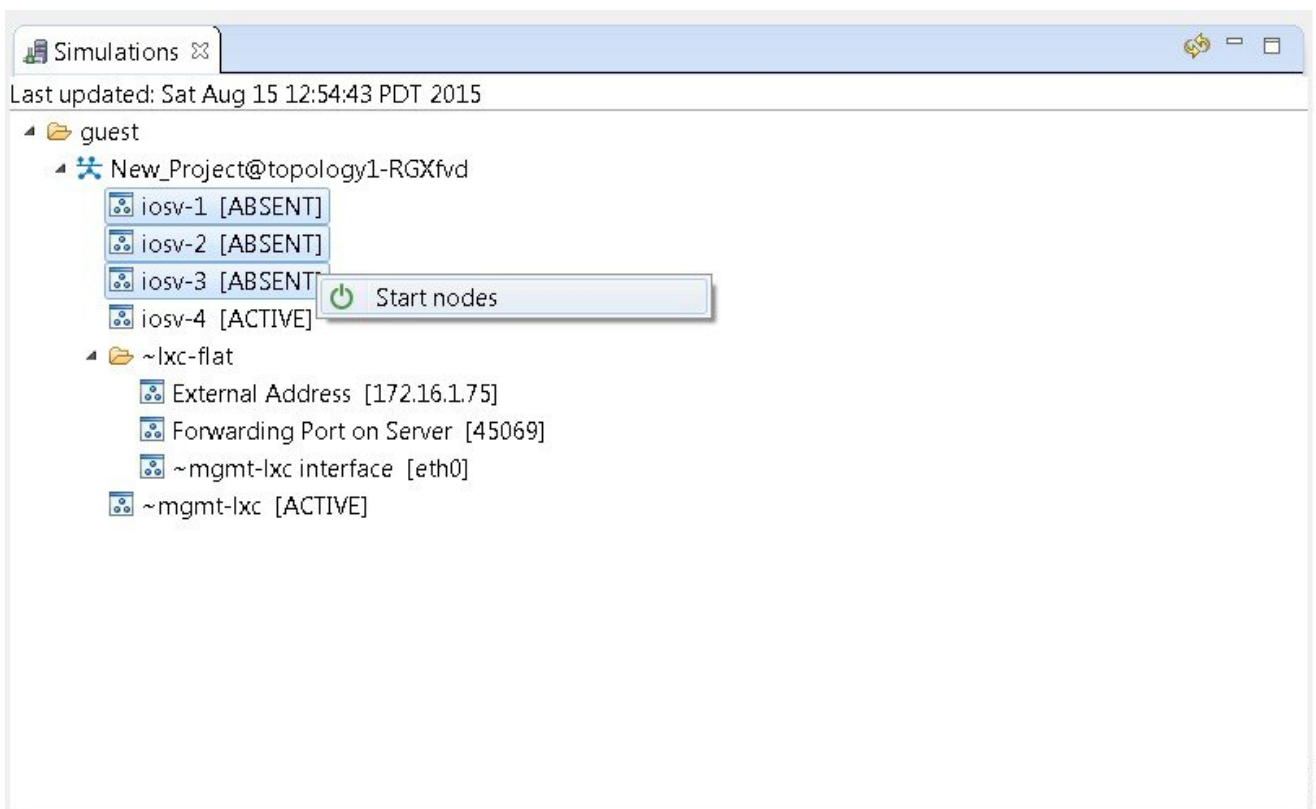
Note The status of the selected node is changed from [ABSENT] to [ACTIVE], indicating that the node is up and running.

Start Multiple Nodes in a Running Simulation

In cases where a phased simulation is running, you can later start those nodes not started with the initial simulation. To start multiple nodes, complete the following steps.

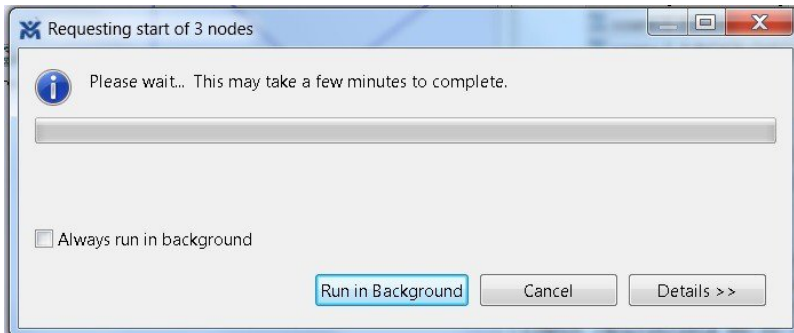
- Step 1** In the **Simulations** view, click the first node in the list to be started.
- Step 2** Hold down the **Shift** key and select the remaining nodes.
- Step 3** Right-click the selected nodes.
- Step 4** Click **Start nodes**.

Figure 30: Start Nodes Option



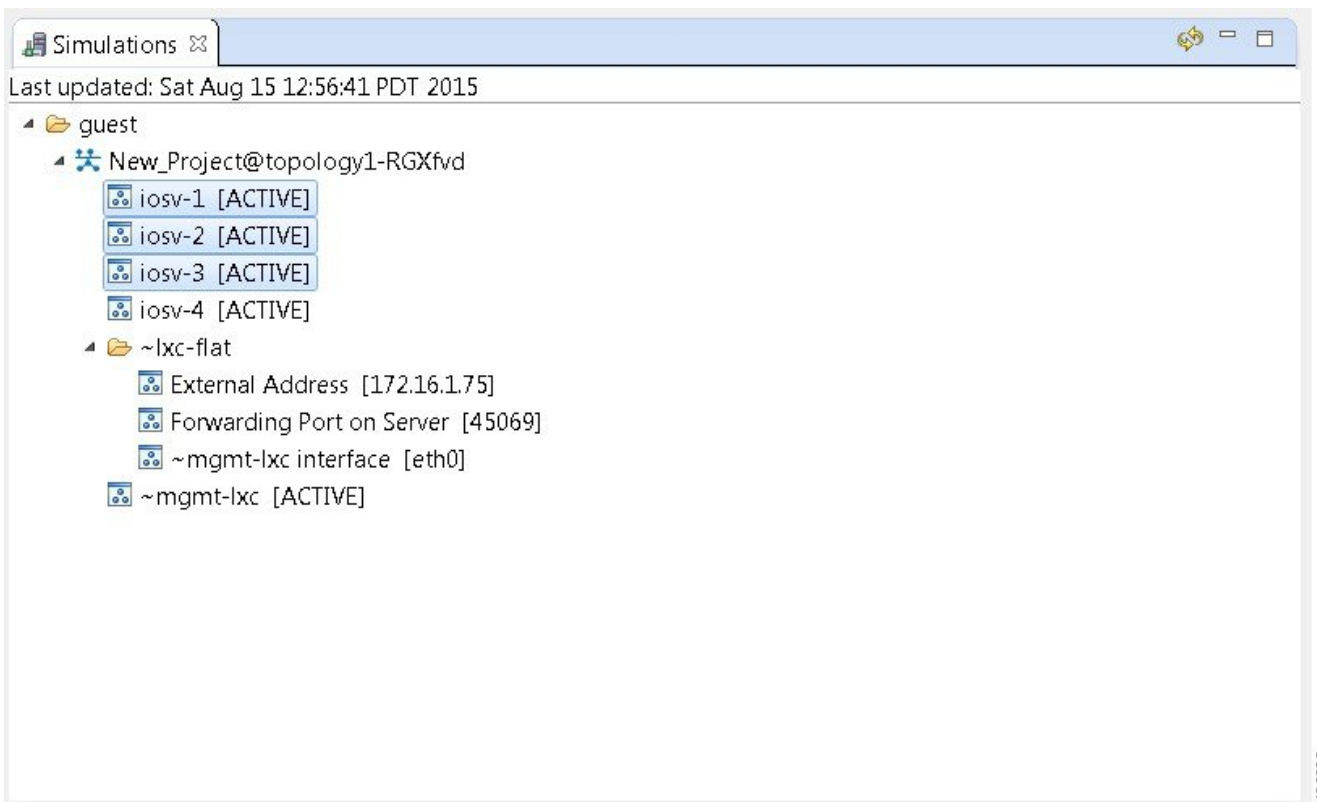
The **Requesting start** dialog box appears.

Figure 31: Requesting Start Dialog Box



Note The status of the selected nodes changes from [ABSENT] to [ACTIVE], indicating that the nodes are up and running.

Figure 32: Multiple Nodes Started



Stop a Simulation

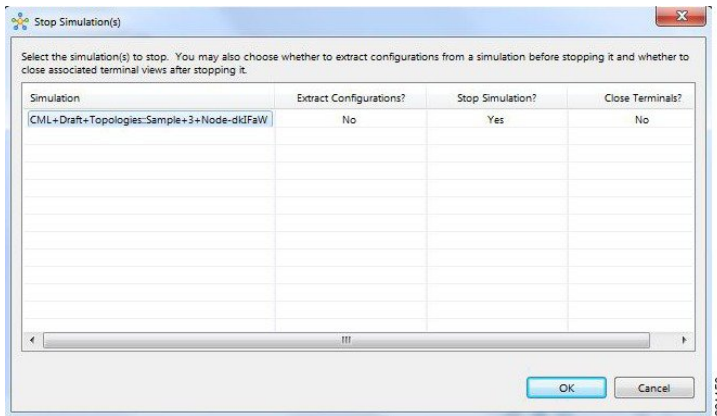
There are several ways to stop a simulation. In addition, you can stop multiple simulations at the same time. These are discussed in the following sections.

Stop a Simulation from the Toolbar

To stop a simulation from the toolbar, complete the following steps.

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

Figure 33: Stop Simulation(s) Dialog Box



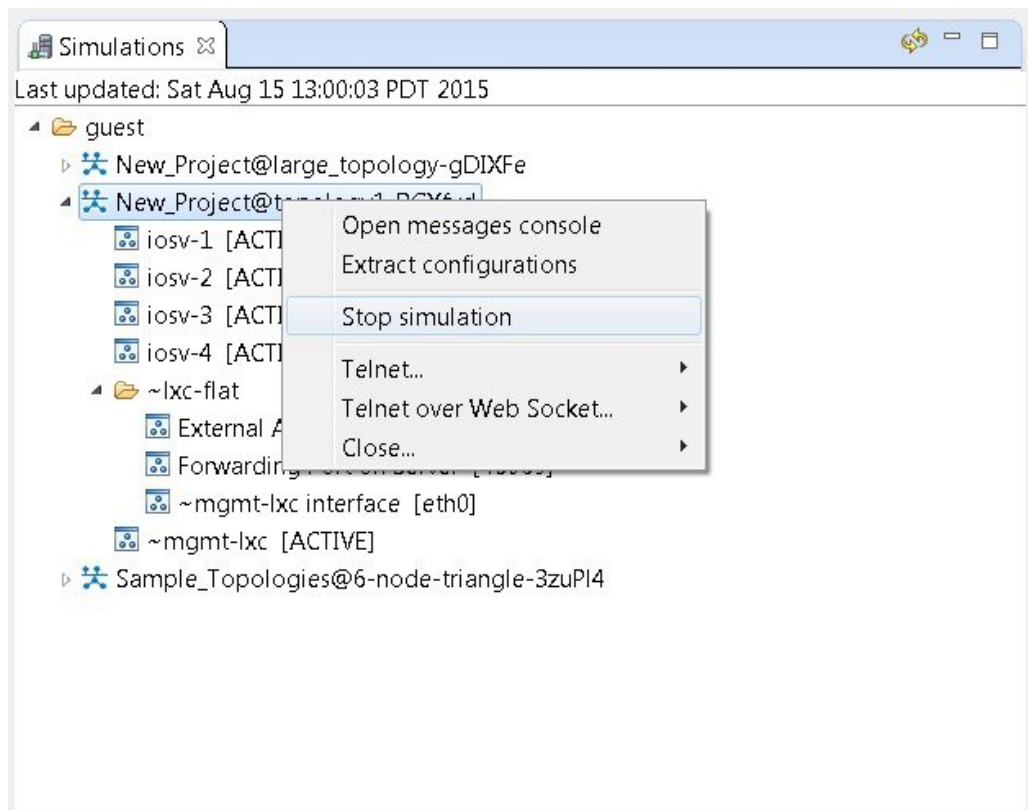
- Step 2** In the **Simulation** column, click once to highlight the simulation to stop.
- Step 3** (Optional) To save the configurations, click the adjacent setting in the **Extract Configurations?** column until the prompt changes to **Yes**.
Note Configurations for server nodes are not extracted.
- 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 stop the simulation, or click **Cancel** to leave the simulation running. On 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.

Stop a Simulation from the Simulations View

To stop a simulation, complete the following steps.

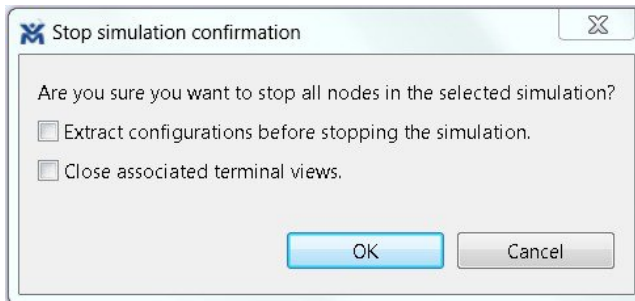
Step 1 In the **Simulations** view, right-click the simulation name and select **Stop Simulation**.

Figure 34: Stop a Simulation



The **Stop Simulation Confirmation** dialog box appears.

Figure 35: Stop Simulation Confirmation Dialog Box



- (Optional) Check the **Extract Configurations before Stopping the 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.

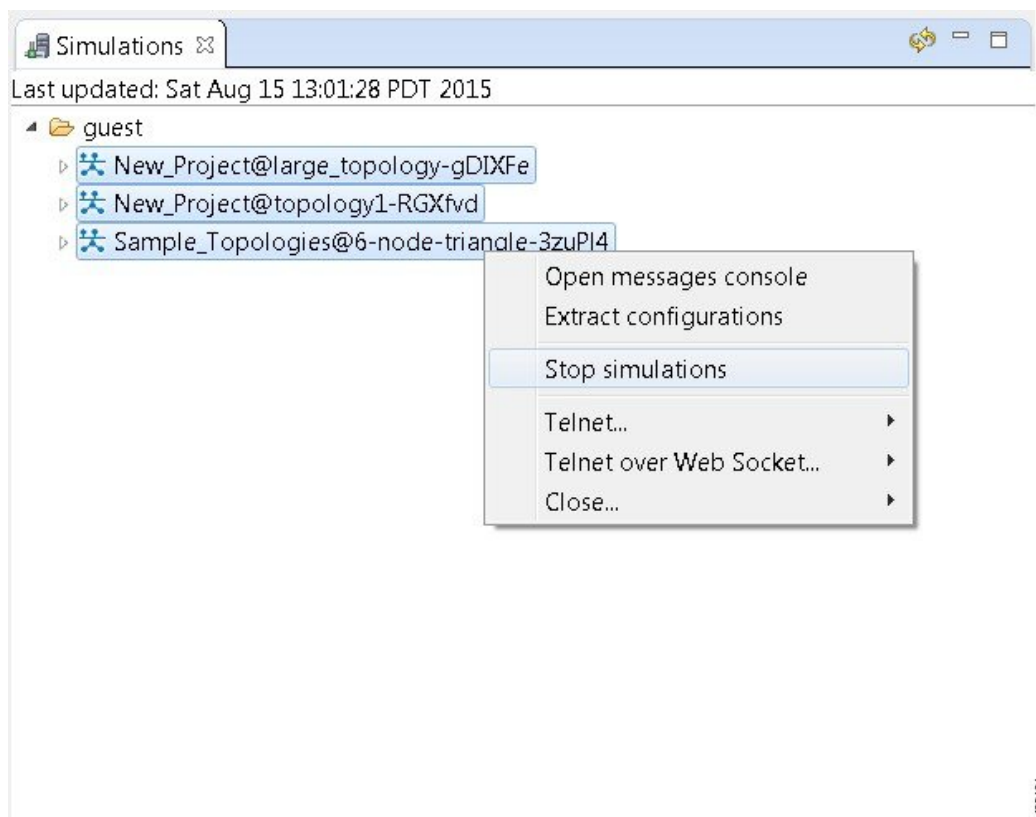
Note For instances where a user account expires, all running simulations for that user continue to run. Since the user account can no longer log in to stop them, they will remain active until the next system reboot or until the system administrator explicitly stops them.

Stop Multiple Simulations from the Simulations View

To stop multiple simulations, complete the following steps.

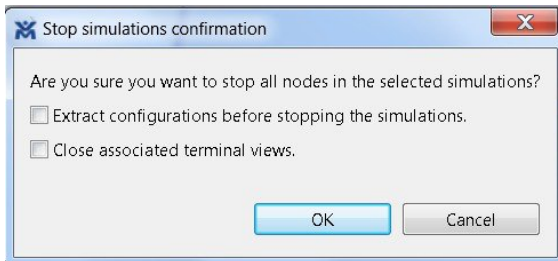
- Step 1** In the **Simulations** view, click the first simulation in the list to stop.
- Step 2** Hold down the **Shift** key and select the remaining simulations.
- Step 3** Right-click the selected simulations and select **Stop Simulations**.

Figure 36: Stop Multiple Simulations



The **Stop Simulations Confirmation** dialog box appears.

Figure 37: Stop Simulations Confirmation Dialog Box



- (Optional) Check the **Extract Configurations before Stopping the 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 4 Click **OK** to stop the simulations.

Once selected, all nodes in the simulations start shutting down. It may take a few minutes for the simulations to shut down completely and to disappear from the **Simulations** view.

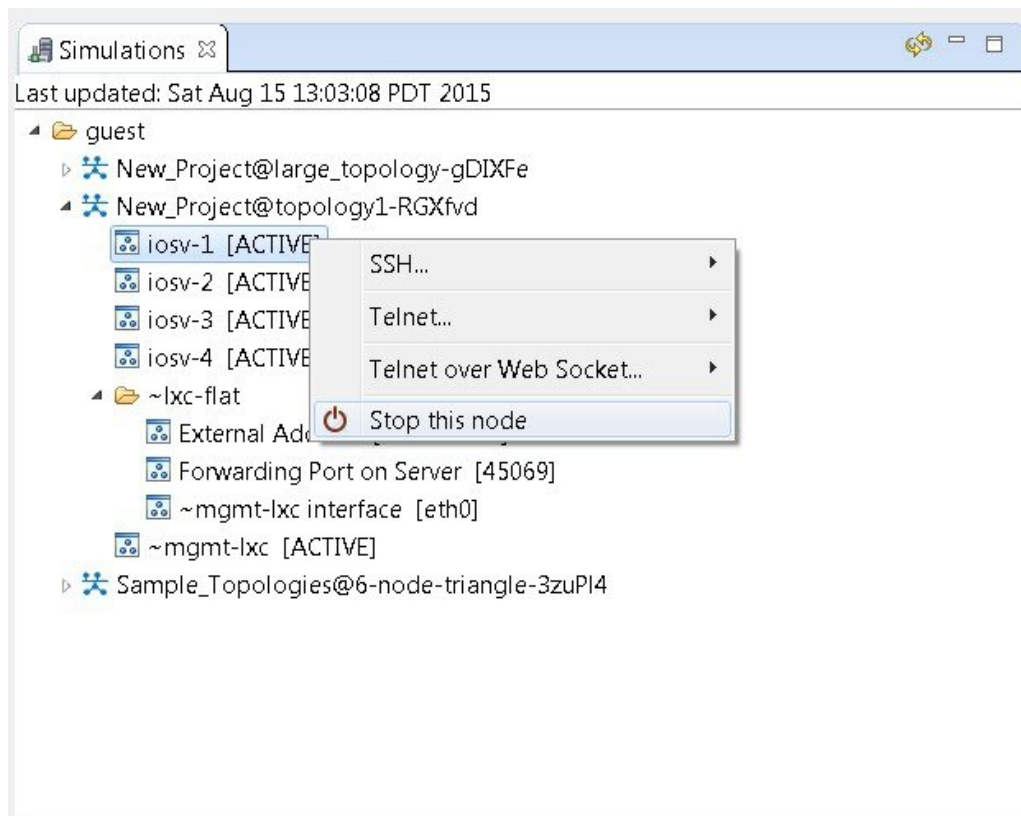
Note For instances where a user account expires, all running simulations for that user continue to run. Since the user account can no longer log in to stop them, they will remain active until the next system reboot or until the system administrator explicitly stops them.

Stop a Single Node

To stop a single node in a simulation, complete the following steps.

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

Figure 38: Stop a Single Node



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

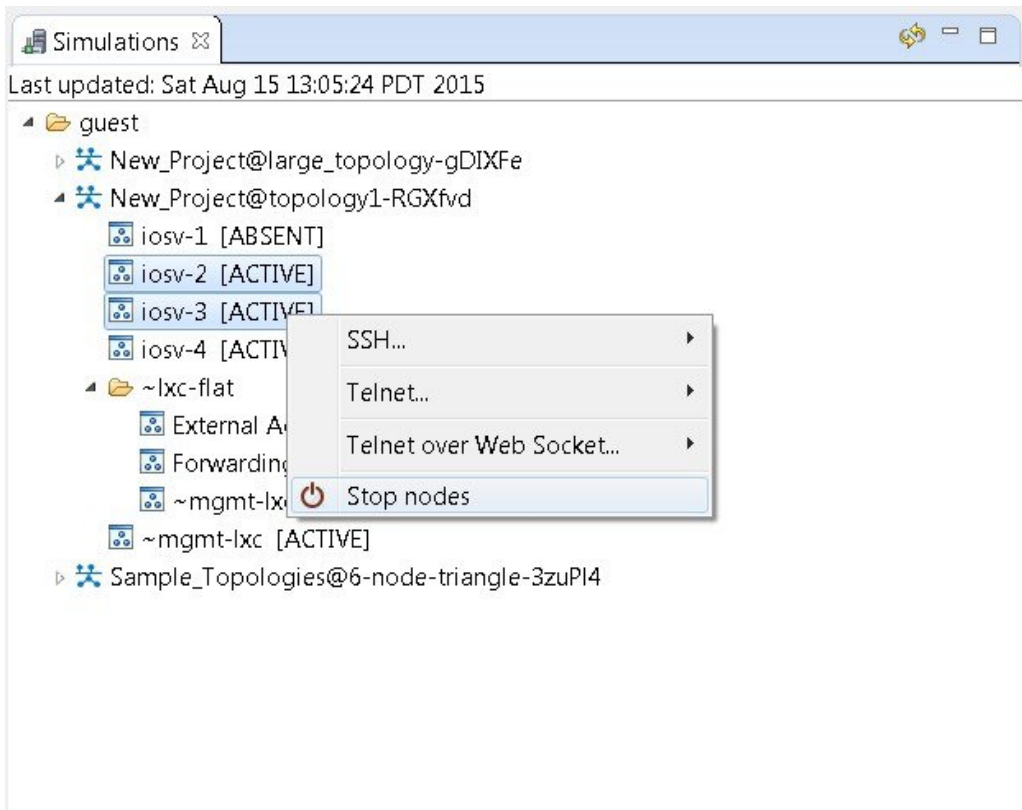
- Step 2** Click **OK** to stop the node. Alternatively, click **Cancel** to abandon the operation and return to the simulation.
- Note** When you click **OK**, the node stops without saving any changes to the configuration.
- When a node is stopped, its status changes to **[ABSENT]**.

Stop Multiple Nodes

To stop multiple nodes in a running simulation, complete the following steps.

- Step 1** In the **Simulations** view, click the first node in the list to stop.
- Step 2** Hold down the **Shift** key and select the remaining nodes.
- Step 3** Right-click the selected nodes.
- Step 4** Click **Stop Nodes**.

Figure 39: Stop Multiple Nodes

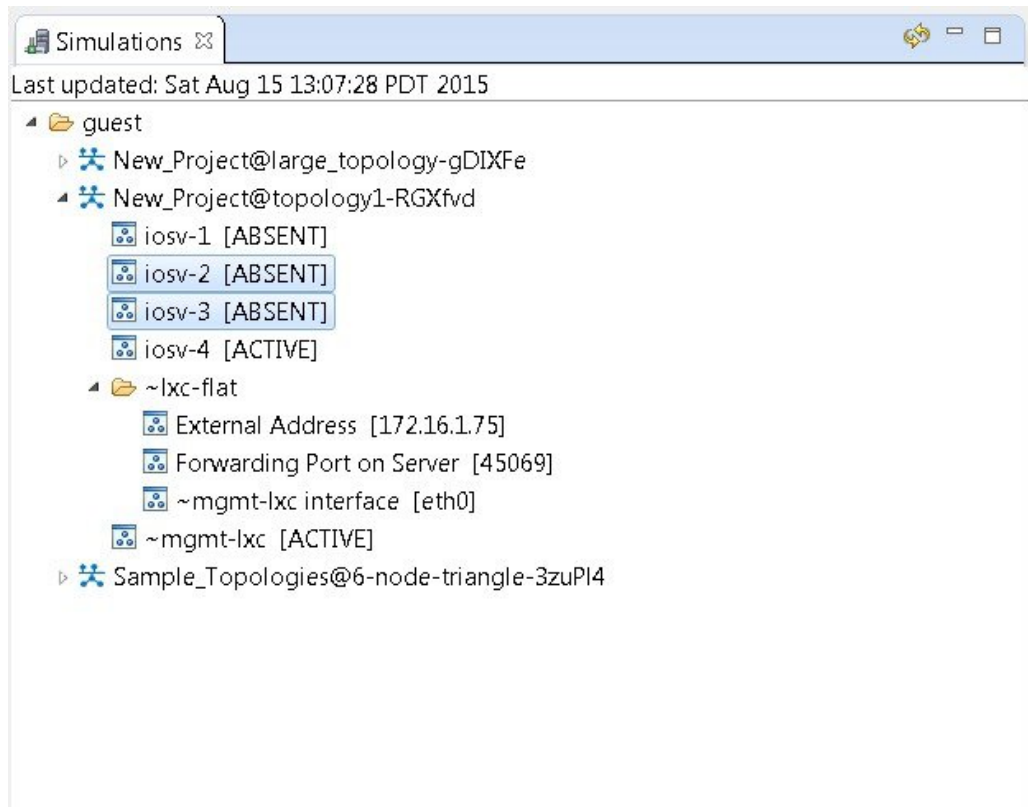


The **Are you sure?** dialog box appears.

- Step 5** Click **OK** to stop the nodes. Alternatively, click **Cancel** to abandon the operation and return to the simulation.
- Note** When you click **OK**, the nodes stop without saving any changes to the configuration.

When the nodes are stopped, their status changes to **[ABSENT]**.

Figure 40: Multiple Nodes Stopped



Modify a Node Configuration in the Simulation

You can modify node configurations in a running simulation. To do this, the type of connection available, either SSH or Telnet depends on the option set for the topology property **Management Network**.

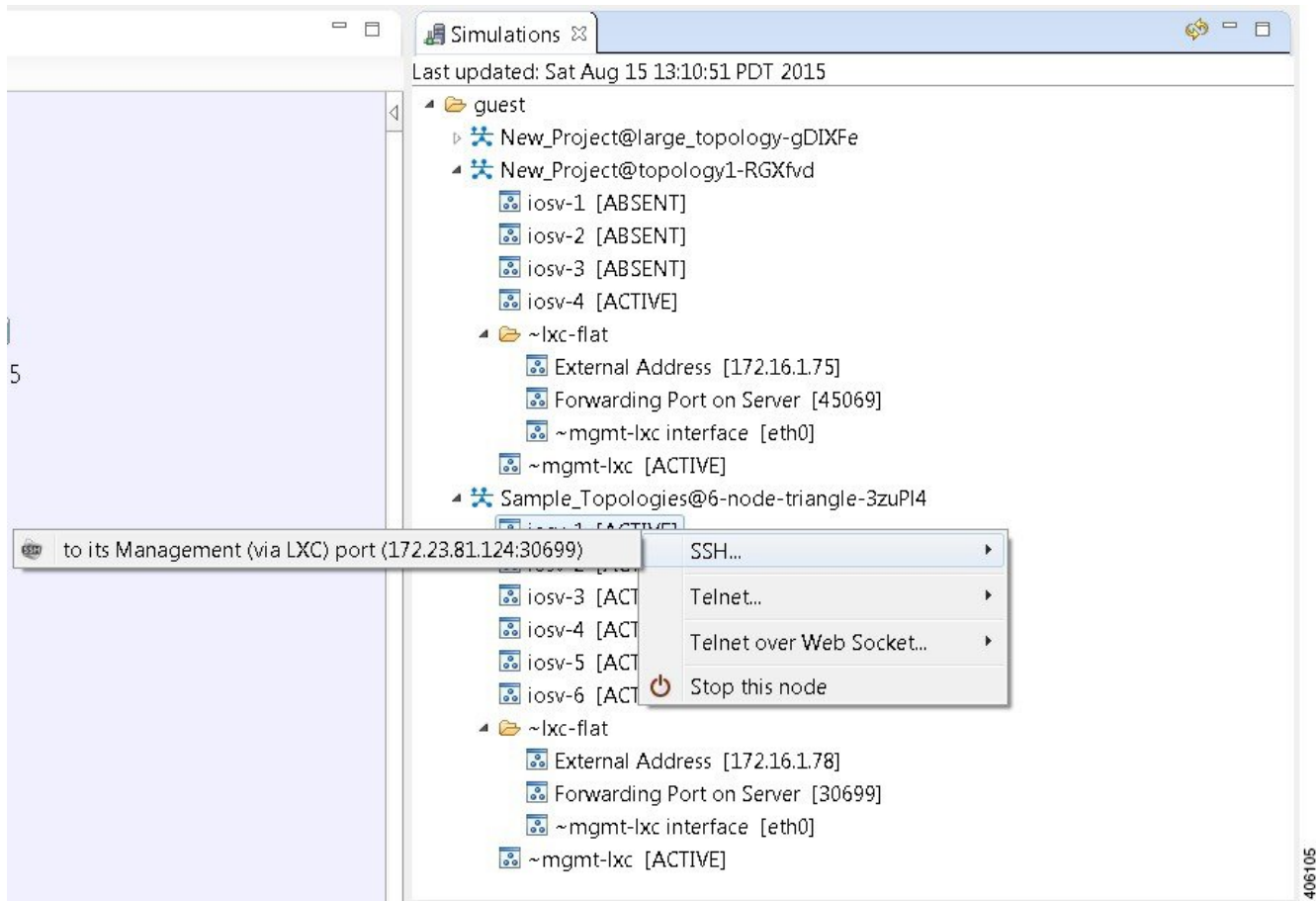
- When **Management Network** is set to **Private simulation network**, SSH and Telnet are available.
- When **Management Network** is set to **Shared flat network**, **Private project network**, or **not specified**, only Telnet is available.

Modify a Node Configuration in the Simulation via SSH

To modify a node configuration in a running simulation via SSH, complete the following steps.

Step 1 Right-click the node in the **Simulations** view and choose **SSH > to its Management (via LXC) port**.

Figure 41: Connecting to a Node Console



A new **Terminal** view opens.

Figure 42: Terminal View

```

Console Cisco Terminal 1 iosv-1 (Management (via LXC)) - guest.Sample_Topologies@6-node-tri...
SSH: (guest@172.23.81.12430757 - CONNECTED)
Connected to 10.255.0.4.
Escape character is '^]'.

*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing. *
*****

User Access Verification

Password:
*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing. *
*****

iosv-1>
iosv-1>

```

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 IOSv virtual software.

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

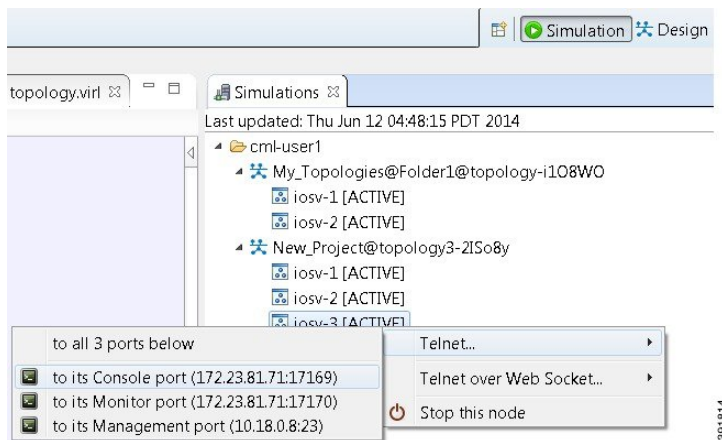
Note Changes you make to the configuration do not appear in the canvas of the **Topology Editor**.

Modify a Node Configuration in the Simulation via Telnet

To modify a node configuration in a running simulation via Telnet, complete the following steps.

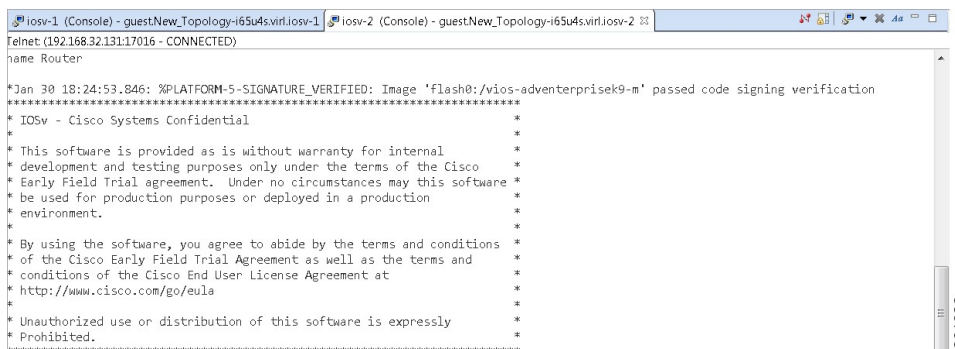
Step 1 Right-click the node in the **Simulations** view and choose **Telnet > to its Console port**.

Figure 43: Connecting to a Node Console



A new **Terminal** view opens.

Figure 44: 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 IOSv virtual software.

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

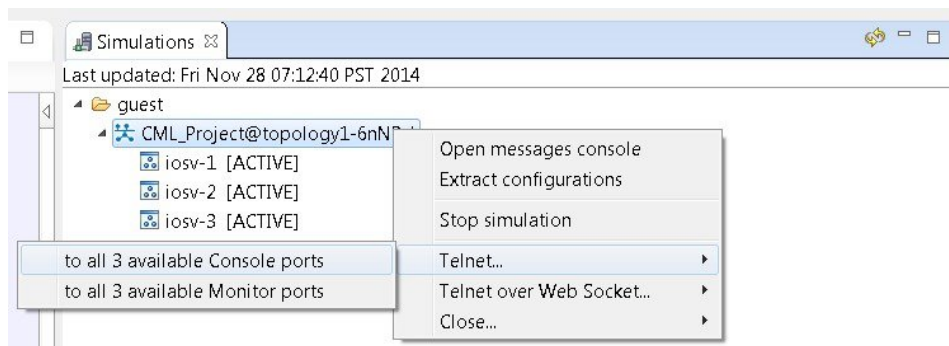
Note Changes you make to the configuration do not appear in the canvas of the **Topology Editor**.

Modify Multiple Node Configurations in the Simulation

To modify multiple node configurations in a running simulation, complete the following steps.

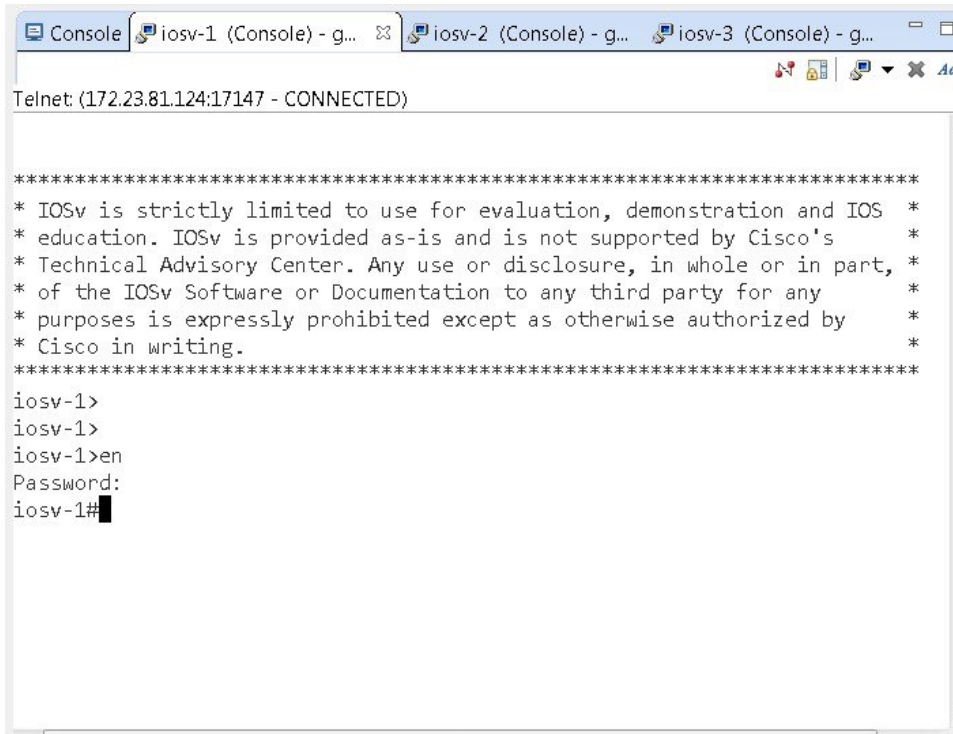
Step 1 Right-click the topology in the **Simulations** view and choose **Telnet > to all <number> available Console ports**.

Figure 45: Connect to Multiple Node Consoles



A new **Terminal** view opens for each of the consoles.

Figure 46: Terminal Views



```

Telnet: (172.23.81.124:17147 - CONNECTED)

*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing.
*****

iosv-1>
iosv-1>
iosv-1>en
Password:
iosv-1#

```

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 IOSv virtual 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

To extract and save modified configurations, complete the following steps.

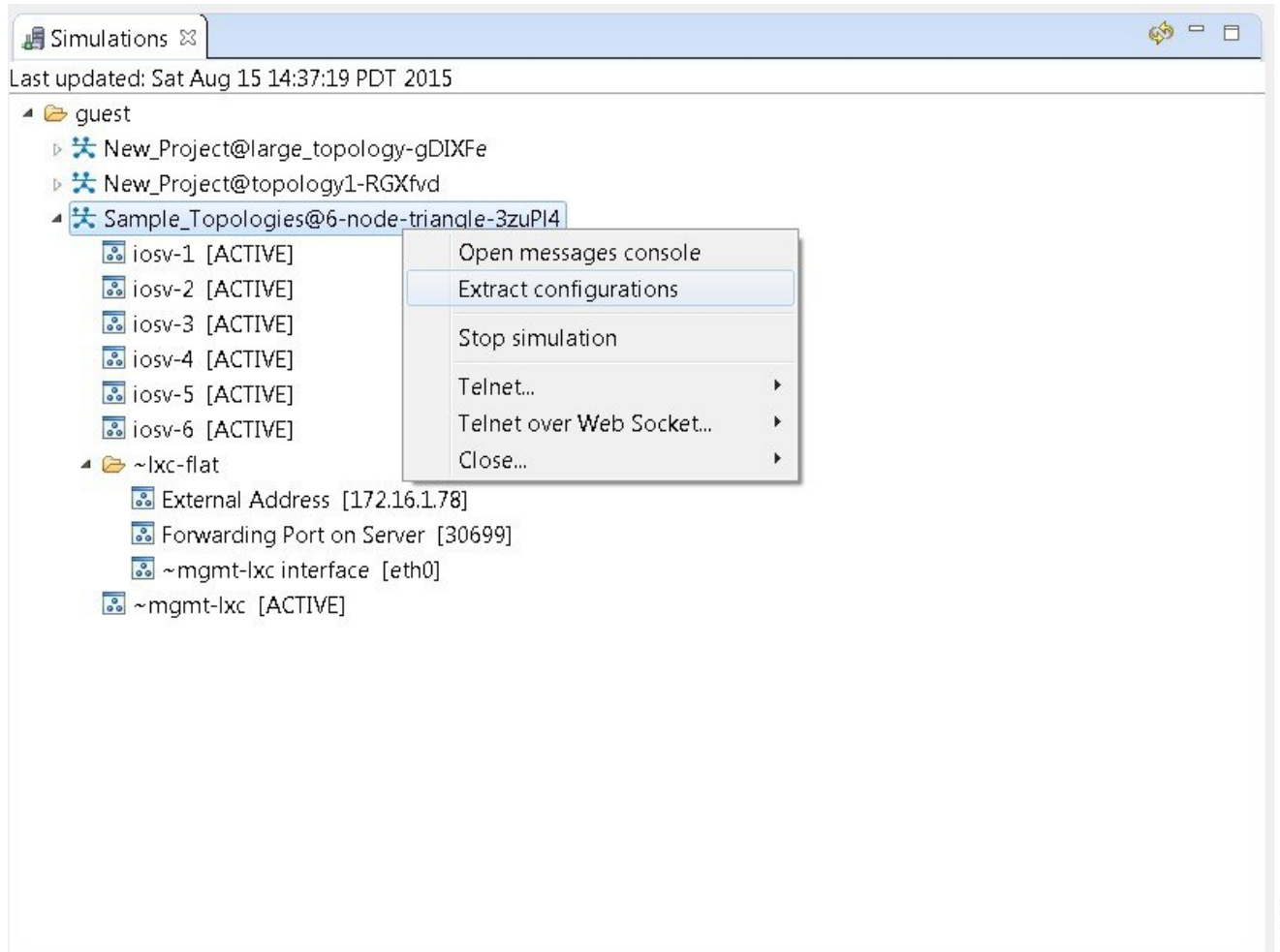
Before You Begin

- 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 47: Simulations View Context Menu

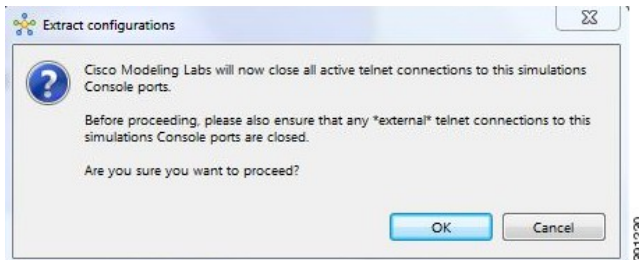


A confirmation dialog box appears.

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 48: Extract Configurations Dialog Box



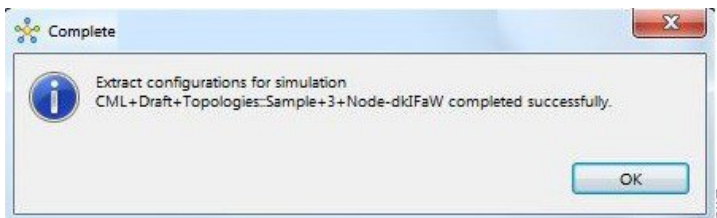
Step 3

Click **OK**.

The Extracting Configurations dialog box appears indicating that the extraction is in process.

When the extraction is complete, a message is displayed.

Figure 49: Extraction Complete Message



Note The configuration information is extracted and saved in the filename.virl 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 4

Click **OK**.

Partial Configuration Extraction

During a configuration extraction, if the process encounters issues or fails for a particular node, the problem node is identified and reported.

The extraction process then continues for all other nodes in the simulation and returns collected configurations to you.

Linux Server Snapshot Support

When a Linux server is present in a running simulation, you can use the **User Workspace Management** interface to take a snapshot of the disk content of the server. This newly created user-specific disk image can be used in other simulated sessions.

To take a snapshot of the server's disk contents, complete the following steps.

- Step 1** Log in to the **User Workspace Management** interface.
- Note** You must log in as a user other than the uwmadmin user, for example, guest.
- Step 2** On the **Overview** page, under **Sessions**, select the applicable running simulation.

Figure 50: Running Simulation Listed

User Workspace Management Logged in as guest Log out

Sessions

Session	Status	Expires	Options
~jumpshot	STOP	never	<input type="button" value="v"/>
CML_Project@topology1-Xc7wLD	ACTIVE	never	<input type="button" value="v"/>

Usage statistics for user guest

4 / 100 instances

1552 / 512000 RAM (MB)

4 / 200 VCPUS

A list of active VMs is displayed.

Figure 51: Active VMs

Session CML_Project@topology1-Xc7wLD details

- Sessions
- Project sessions
- Images
- Flavors
- Subtypes
- Connectivity
- Settings and password

Nodes

Node	Subtype	Status	Options
iosv-1	IOSv	ACTIVE	<input type="button" value="v"/>
iosv-2	IOSv	ACTIVE	<input type="button" value="v"/>
server-1	server	ACTIVE	<input type="button" value="v"/>
~mgmt-kxc	mgmt-kxc	ACTIVE	

Interfaces

Node	Interface name	Network subtype	Network name	IP Addresses
iosv-1	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.3 / 16
iosv-2	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.1 / 16
server-1	eth0	SESSION MGMT	mgmt	10.255.0.4 / 16
~mgmt-kxc	eth0	LXC FLAT	flat	172.16.1.147 / 24; public port: 30799
~mgmt-kxc	eth1	SESSION MGMT	mgmt	10.255.0.2 / 16

Step 3 Select the applicable Linux server, and from the **Options** drop-down menu, click **Create snapshot**.

Figure 52: Create Snapshot Option

Session CML_Project@topology1-Xc7wLD details

Sessions

Project sessions

Images

Flavors

Subtypes

Connectivity

Settings and password

Nodes

Node	Subtype	Status	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
server-1	server	ACTIVE	▼
~mgmt-lxc	mgmt-lxc	ACTIVE	<ul style="list-style-type: none"> ▣ VNC console ▸ Serial port 📄 Create snapshot

Interfaces

Node	Interface name	Network subtype	Network name	IP Addresses
iosv-1	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.3 / 16
iosv-2	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.1 / 16
server-1	eth0	SESSION MGMT	mgmt	10.255.0.4 / 16
~mgmt-lxc	eth0	LXC FLAT	flat	172.16.1.147 / 24; public port: 30799
~mgmt-lxc	eth1	SESSION MGMT	mgmt	10.255.0.2 / 16

Project details for the newly created snapshot are displayed.

Figure 53: Newly Created Disk Image

User Workspace Management Logged in as guest Log out

Saving snapshot of node "server-1" as image "guest-server-server-1"...

Project image snapshot guest-server-server-1 details

Name
guest-server-server-1

OpenStack ID
e5da2898-4c68-4d3c-8a8b-0a649f905a8a

Project
d14434d936614631a0055da7deae8cca

Updated
2014-12-15 13:34:07

Status
queued

Size
0.00 B (0)

Minimum Disk Size
undefined

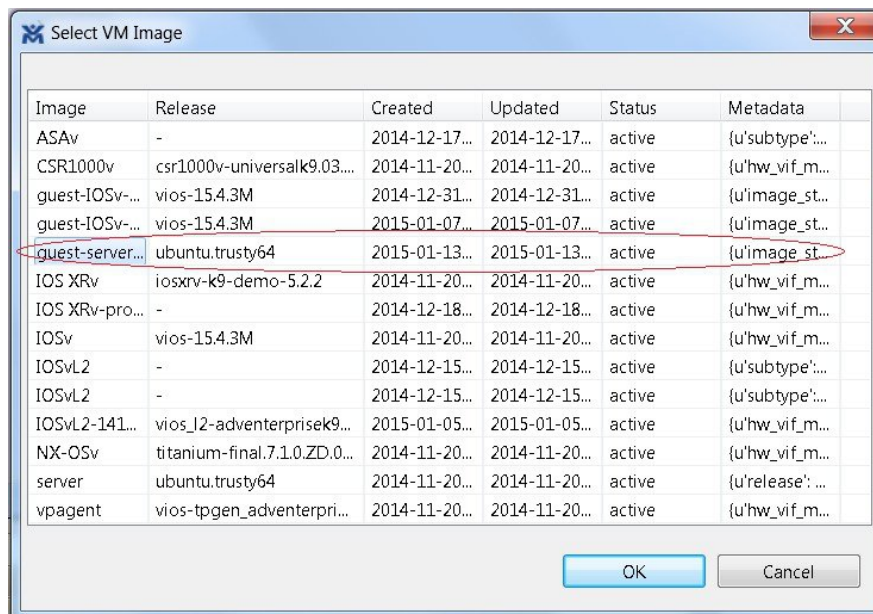
Converted qcow2 image checksum
None (every image is forced through conversion, even qcow2)

Reuse the Image Snapshot

To reuse the image snapshot, complete the following steps.

- Step 1** Create a new topology or open an existing topology.
- Step 2** On the canvas, add a node to the topology.
- Step 3** Select the node on the canvas.
The sample topology opens in the **Topology Editor** canvas.
- Step 4** In the **Properties > Node** view, click **Browse** beside the **VM Image** field.
The **Select VM Image** dialog box appears.
- Step 5** Select the applicable image snapshot and click **OK**.

Figure 54: Select the Image Snapshot to Use



Details for the image snapshot are visible in the **VM Image** field under **Properties > Node** view.

Figure 55: Image Snapshot Selected

