



Design a Topology

- [Design a Topology Overview](#), page 1
- [Topology Nodes and Connections](#), page 1
- [Create a Topology Project](#), page 3
- [Create a Topology](#), page 5
- [Place the Nodes on the Canvas](#), page 6
- [Create Connections and Interfaces](#), page 7
- [Create Multipoint Connections](#), page 8
- [Create a Site](#), page 9
- [View the Topology on a Map](#), page 10

Design a Topology Overview

The design phase is the initial step in creating a network topology. During the design phase, you will perform the tasks described in the following sections.

Topology Nodes and Connections

The topology you design consists of nodes and connection functions. Refer to the section [Navigating Within the Cisco Modeling Labs Client](#) for additional information about how to select and edit nodes and connection functions.

Topology Nodes

Cisco Modeling Labs provides the Cisco IOSv node subtype. Additional node types are available and can be installed by the system administrator via the User Workspace Management interface. See the User Workspace Management chapter in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide*.

Table 1: Node Subtypes

Node Name	Node Type
Cisco IOSv	Router node. Runs a Cisco IOS operating system.
Server	Server node. Runs a Linux operating system.
Cisco IOS XRv	Router node. Runs a Cisco IOS XR operating system. (Available separately.)
Cisco CSR1000v	Router node. Runs a Cisco CSR 1000 operating system. (Available separately.)

A node subtype is a virtual machine that runs on top of OpenStack, which itself is running in a Linux virtual machine that is running on top of VMware software. Because the node is virtual, specific hardware is not emulated. For example, there are no power supplies, no fans, no ASICs, and no physical interfaces. For all router nodes, the interface type is a Gigabit Ethernet network interface. A server node has an Ethernet network interface.

You can choose an image and image flavor for each node type. Refer to the User Workspace Management chapter in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide* for information on how to access the VM Image and the VM Flavor choices. In most cases, you need not select an image and flavor. By default, the node subtype is associated with an image and flavor that runs with the topology.

Table 2: Node VM Images

VM Image Name	Used For
server	Server node
CSR1000v	Cisco CSR1000v node
IOSv	Cisco IOS node
IOS XRv	Cisco IOS XR node

Table 3: Node VM Flavors

VM Flavor Name	Used For
m1_tiny	Linux server
m1_small	Linux server
m1_medium	Linux server
m1_large	Linux server
m1_xlarge	Linux server
server	Linux server
CSR1000v	Cisco CSR 1000v node

VM Flavor Name	Used For
IOS XRv	Cisco IOS XR node
IOSv	Cisco IOS node

Each Linux flavor provides a different amount of memory and CPU allocated to the server.

Connection Functions

Cisco Modeling Labs provides the connection functions shown in the following table.

Table 4: Connection Functions

Connection Type	Description
Connection	Creates a connection between two interfaces. Interfaces are created in the node to support a connection. Any unused interfaces present are automatically assigned. All the interfaces in router nodes are represented as Gigabit Ethernet interfaces. Multiple parallel connections are supported.
Multipoint Connection	Creates a multipoint connection point. Multiple nodes can connect to a multipoint connection.
Layer 3 External (SNAT)	Creates a Layer 3 external connection point using static network address translation (SNAT). This external connection point allows connections outside of Cisco Modeling Labs to connect to the topology.
Layer 2 External (Flat)	Creates a Layer 2 external connection point using FLAT. This external connection point allows connections outside of Cisco Modeling Labs to connect to the topology.

Table 5: Site Groups

Type	Description
Site	Creates a site that can contain one or more nodes, and additional groups of nodes. A site is a group of nodes and you can create multiple site layers.

Create a Topology Project

Cisco Modeling Labs client provides a project folder and a sample topology that you can work with in addition to creating your own project and topology.

A topology project is a folder in which multiple topology files are stored. If you create many different topologies, you can set up different projects for the topologies you create.

The three methods available for creating a topology project are discussed in the following sections.

Before You Begin

Before you begin the task of creating a topology project, you must complete the installation steps for your system and set up the web services profile for the Cisco Modeling Labs client. Refer to the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide*. See [Navigating Within the Cisco Modeling Labs Client](#) for information on how to navigate the perspectives and views, and information about the menu bars, toolbars, and editors. The navigation information is also available in the online help.

In addition, understand how:

- To navigate Cisco Modeling Labs client.

Method 1: Create a Topology Project from the Menu Bar

- Step 1** From the menu bar, choose **File > New > Other > Topology Project**.
- Step 2** Enter a Project Name and select the location in which to save the folder. A project folder is created in the default location if you check the **use default location** check box. Alternatively, you can choose to save the project folder in another location.
- Step 3** Click **Finish**.
-

Method 2: Create a Topology Project from the Projects View

- Step 1** Right-click **Projects** view.
- Step 2** Choose **New > Topology Project**.
- Step 3** Enter a Project Name and select the location in which to save the folder. A project folder is created in the default location if you check the **use default location** check box. Alternatively, you can choose to save the project folder in another location.
- Step 4** Click **Finish**.
-

Method 3: Create a Topology Project from the Toolbar

- Step 1** Click the **New Topology Project** icon in the toolbar.
- Step 2** Enter a Project Name and select the location in which to save the folder. A project folder is created in the default location if you check the **use default location** check box. Alternatively, you can choose to save the project folder in another location.
- Step 3** Click **Finish**.
-

What to Do Next

Create a new topology file.

Create a Topology

Before You Begin

A topology project folder must exist.

There are several methods for creating a topology. These are discussed in the following sections.

Method 1: Create a Topology from the Menu Bar

- Step 1** From the menu bar, choose **File > New > Other > Topology**.
- Step 2** Select a topology project folder.
- Step 3** Enter a filename, ensuring that it ends with the extension **.virl**.
- Step 4** Click **Finish**.
A *filename.virl* topology file is created in the selected project folder.
-

Method 2: Create a Topology from the Projects View

- Step 1** Right-click **Projects** view.
- Step 2** Choose **New > Topology**.
- Step 3** Select a topology project folder.
- Step 4** Enter a filename, ensuring that it ends with the extension **.virl**.
- Step 5** Click **Finish**.
A *filename.virl* topology file is created in the selected project folder.
-

Method 3: Create a Topology from the Toolbar

- Step 1** Click the **New Topology File** icon in the toolbar.
- Step 2** Choose **New > Topology**.
- Step 3** Select a topology project folder.
- Step 4** Enter a filename, ensuring that it ends with the extension **.virl**.
- Step 5** Click **Finish**.
A *filename.virl* topology file is created in the selected project folder.
- Note** *filename.virl* is a placeholder for example purposes only.
-

What to Do Next

Place the nodes.

Place the Nodes on the Canvas

Before You Begin

- A topology file must exist.
- The topology file must be open and the canvas visible in the Topology Editor.

-
- Step 1** Click a node type, which is under the Nodes heading in the **Palette** view.
- Step 2** Click the canvas at each point where you want a node. You can also drag the nodes on the canvas to position them. You can then arrange the nodes using several methods:

- Use Shift-click to select two or more nodes. Alternatively, click and drag a selection box around two or more nodes. You can also use Ctrl+click (Windows) or Cmd+click (Mac OS X) to toggle between node selections.
- Use the **Distribute Nodes** tool on the toolbar to arrange the selected nodes vertically, horizontally, or to distribute them on a grid.
Note The grid is determined dynamically, based on the selected nodes.
- Use the **Align Nodes** tool on the toolbar to arrange the selected nodes in a variety of alignments that you choose from the **Alignment** drop-down list.
Note The use of **Distribute Nodes** and **Align Nodes** is mutually exclusive.
- When no nodes are selected, use the **Layout Nodes** tool on the toolbar to arrange the nodes in a Tree Layout or a Fruchterman-Reingold (F-R) Layout.
 - A basic tree layout is to choose a node as the root of the tree, and the nodes connected to the root become children of the root node.
 - A F-R layout has connected nodes close to each other, while unconnected nodes are further apart from each other.

See the section [Toolbar](#) for information on using these tools.

What to Do Next

Create connections and interfaces.

Create Connections and Interfaces

Before You Begin

Nodes must be in place on the canvas of the Topology Editor.

Step 1 Click **Connection** in the **Tools** view.

Step 2 Click the first node.

Step 3 Click the next node to create the connection.

Note When you connect two nodes, the interfaces are created and named automatically when you choose **File > Preferences > Topology Editor > Associate new connection with interfaces silently** check box. You can view the node interfaces when you click the connection and the two end points of the connection appear in the **Node Editor**.

Step 4 Repeat Step 2 and Step 3 until all the connections are in place.

Tip You can create multiple parallel connections between two nodes. When a connection is selected, the **Topology Editor** shows the connection end points and the **Node Editor** shows the two nodes involved in the connection, all of the interfaces on those nodes, and all the connections between those two nodes.

Note Choose **Export > Export Topology Diagram to Image** to capture an image of the current topology on the canvas.

What to Do Next

Create multipoint connections (optional).

Create Multipoint Connections

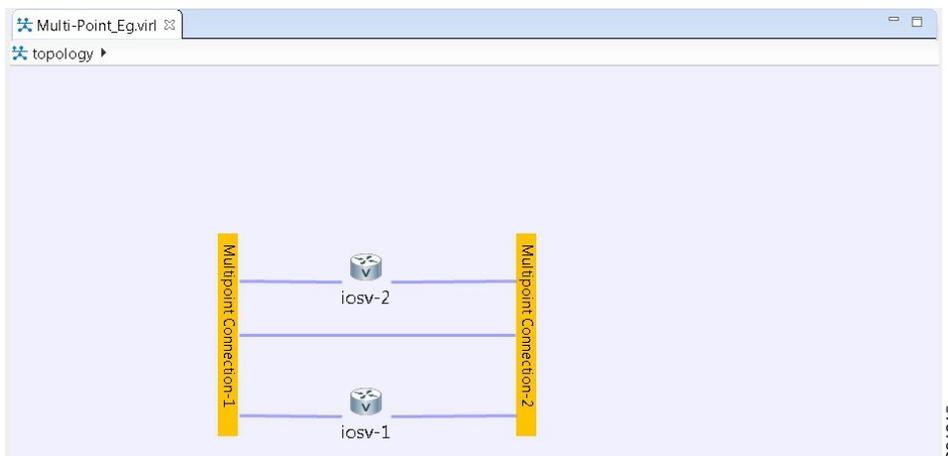
Multipoint connections are used in BUS topologies where all the nodes are connected to a single transmission medium. This acts as the backbone of the connection and links all the nodes in the network. For example, for four nodes to be directly connected to one another, 12 separate interface connections are needed, three in each router, to fully connect each router to the other. By using a multipoint connection, each router has one connection to the multipoint node.

Before You Begin

A topology file with the extension `.virl` must exist. Router nodes or server nodes are placed on the canvas. Optionally, connections may exist between nodes.

- Step 1** In the **General** view, click **Multipoint Connection**.
- Step 2** Click the area on the canvas where you want the multipoint connection node to appear.
- Step 3** In the **Tool** view, click **Connect**.
- Step 4** On the canvas, click the multipoint connection node then click an end node. A connection appears. Continue clicking multipoint-node combinations until all connections are made.
- Note** You can have a connection between two multipoint connection nodes as shown in the following figure.

Figure 1: Creating a Multipoint Connection



- Note** When performing alignment operations for multipoint connections on the canvas of the **Topology Editor**, occasionally, these do not work as expected.

What to Do Next

- Create sites (optional).
- Show map background (optional).

Create a Site

A site is a container that can hold one or more nodes. It is used to group multiple nodes within the same site, which provides a degree of hierarchy and simplifies the topology view.

A node interface within a site can connect to another node interface in the same site or to a node interface located in a different site. A connection cannot start on a node and terminate on a site. Sites can be nested, forming a hierarchical structure.

Before You Begin

One or more nodes should be on the canvas.

Step 1 In the **General** view, click **Site**.

Step 2 Click an area on the canvas to place the site.

Step 3 **Shift**-click the nodes that you want in the site, or click and drag a selection window around the nodes.

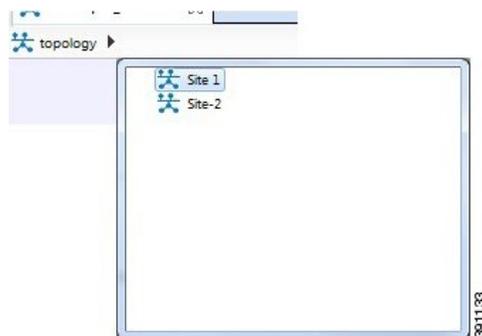
- From the **Edit** menu, choose **Group to Site**.
- Alternatively, you can drag the selected nodes to the site icon that you placed on the canvas.

Step 4 To view the site, select the site name from the drop-down list under the Topology tab. A topology site tab opens and displays the site view.

The drop-down list under the Topology tab displays the site hierarchy as a navigation trail.

To open a site view, double-click on the site. The following figure shows the navigation trail selections for Site 1 and Site 2.

Figure 2: Site Selection



- Step 5** To ungroup the site, click the site icon and choose **Edit > Ungroup Site**.
- Step 6** To connect node interfaces between Site 1 and Site 2:
- Select **Site 1**.
 - Click **Connect**.
 - Click the node that is the starting point of the connection, Site 1.
 - Move the cursor to the breadcrumbs and click **Site 2**.
Site 2 is displayed.
 - Click the node that is the endpoint of the connection.
A connection now exists between the interfaces in the two sites.
- Step 7** To delete a node from a site:
- Select the applicable node.
 - Right-click the node and select **Delete** from the list.
The node is deleted.
- Step 8** To delete a site from a topology:
- Select the applicable site.
 - Right-click the site and select **Delete** from the list.
The site is deleted including all associated interfaces and connections.
-

What to Do Next

Continue designing the topology.

View the Topology on a Map

The topology view is a schematic diagram. It shows the nodes and connections arranged in a logical fashion, or represents a design that is easy to view. When you create a map view, you can move the nodes such that they represent a physical location, for example, the locations of all router nodes in a city. Moving nodes on a map view does not change their location in the topology view.

Within a topology file, each node is represented by two sets of coordinates. The location coordinates define the node placement in the topology view. The map coordinates define node placement on the map view, by latitude and longitude.

The map view uses the MapQuest database for map display.

Before You Begin

- A topology must exist with nodes and connections in place.

- You must be connected to the Internet to use the mapping function, since it uses the map service from MapQuest to retrieve the latest map images.

-
- Step 1** With the topology view active, select **Edit > Show map Background**. Alternatively, you can click the **Show Map Background** tool in the toolbar.
- Step 2** Before you move a node, select the desired map location.
- a) In the **Map** view, click the **Go To Location** icon, located under the pan and zoom controls.
 - Type in the first few characters of a location.
 - On a Windows platform, press Ctrl-Space and a list of matching locations appears.
 - You can enter an airport code.
- Note** We recommend that you enter all upper-case or initial cap words when searching for locations, as these provide more reliable results. For example, enter MEL or Mel to search for Melbourne airport.
- b) Select and click a location from the list.
 - c) Click **OK**.
The map location is displayed.
 - d) Use the pan and zoom controls on the map to determine an exact location. You can double-click the coordinates in the Status bar, which opens the **Go To Location** dialog box.
- Step 3** Click and drag the nodes into position on the map.
When you click and drag a node, the map coordinates are added to the .virl file and the node becomes a part of the map view. When you open the map view again, the map location with the nodes is displayed.
- Step 4** Continue to click and drag the nodes into position on the map view.
- Step 5** Click the **Show Map Background** tool in the toolbar to revert to the topology view. Note that the nodes in the topology view are in their original layout.
-

