# Automate Network Changes

This section contains the following topics:



# Change Automation overview

The Change Automation application automates the process of deploying changes to the network. You can define automation tasks to achieve the intended network states in Change Automation using playbooks that consist of plays written using YAML. You can then push configuration changes to Cisco Network Service Orchestrator (NSO), which deploys these changes to the network devices.

Change Automation, in conjunction with health insights, allows operators to build automation in a *closed-loop framework*. Changes are deployed to the router or other device using programmable APIs, and the intent of the change is verified using telemetry that comes back from the router. Change Automation relies on telemetry to verify the intent of the change, avoiding the need to frequently poll the device for updates.

The following is a high-level Change Automation workflow:

1. Review the existing plays and playbooks to see if they fully or partially meet your needs.

**Note** Change Automation comes with a robust library of playbooks, each with its own collection of configuration and check plays.

2. Build playbook as required:
   - If the required playbook is available, use it.
   - If some combination of existing plays accomplishes the task, build a new playbook using those plays.
   - If some of the required plays are not available, create new plays and build a new playbook using the new and existing plays.

3. Dry run the playbook to test if it performs as expected.
4. Deploy the playbook.

Change Automation allows you to customize and generate plays and playbooks using its API interface. For more information, see About custom plays, on page 5 and About customizing playbooks, on page 12.

# Configure Change Automation settings

Configuring system settings is a post-installation activity and is the first task to be performed after installing Change Automation.

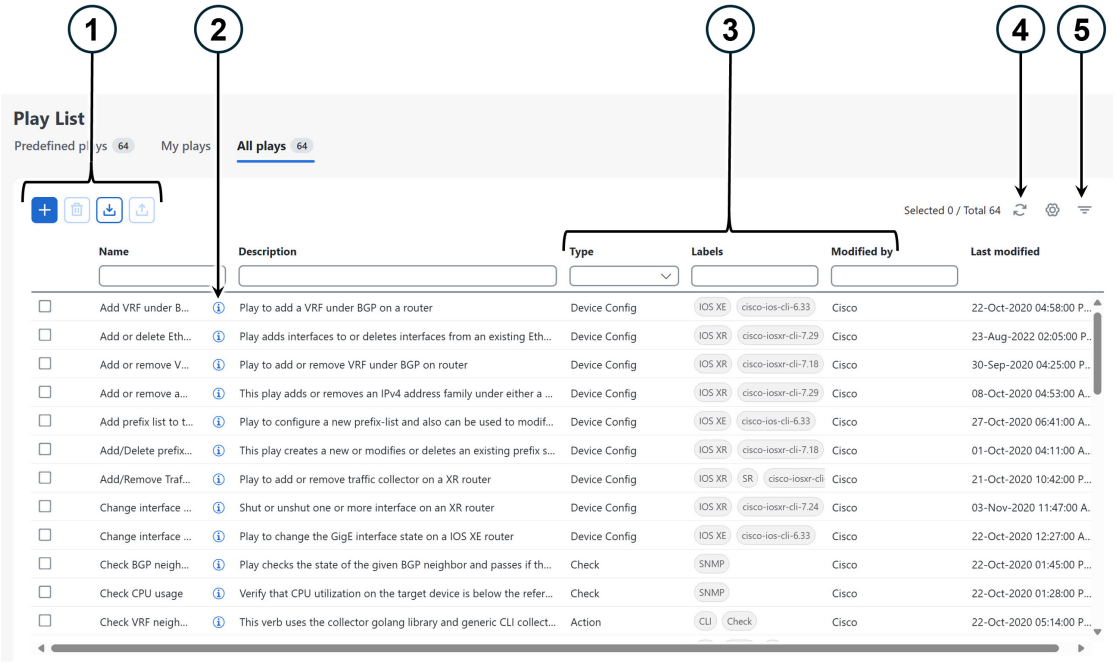
For more information, see Verify installation and configure system settings.

# View the play list

The **Play List** window of the Change Automation application gives you a consolidated list of all the plays in the system.

From the main menu, select **Network Automation** > **Play List** to view the **Play List** window.

***Figure 1: Play List***

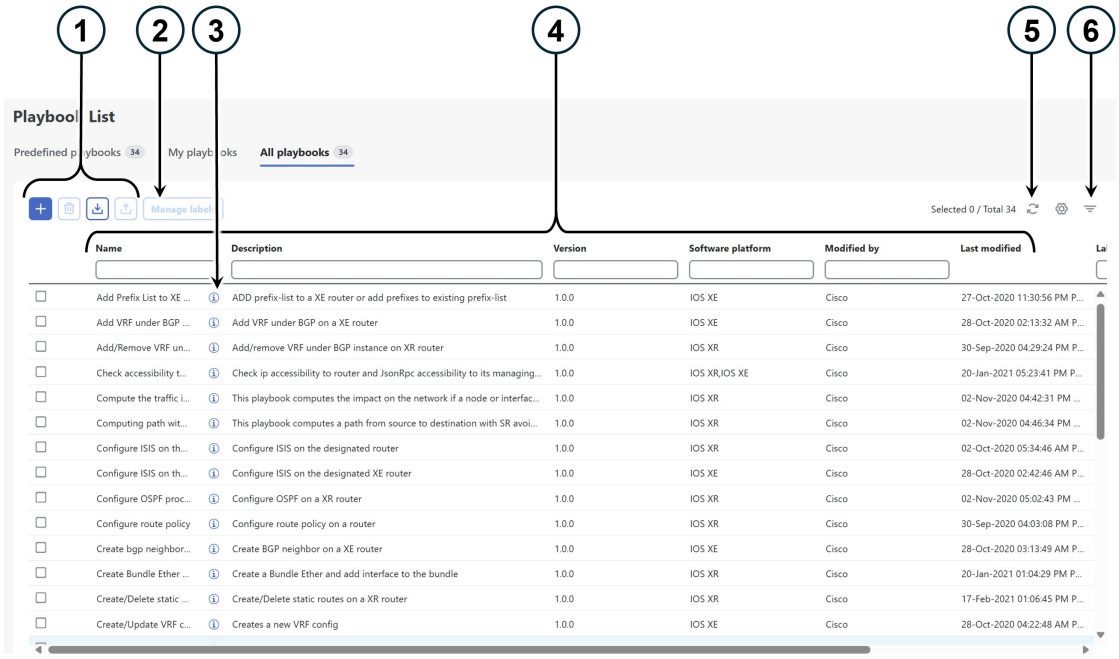


| Item | Description |
|---|---|
| 1 | Click to create a custom play. See Create a custom play using templates, on page 5. |
| | Click to delete a custom play. See Delete custom plays, on page 11. |
| | Click to import a custom play from a gzipped TAR archive file. See Import custom plays, on page 10. |
| | Click to export a custom play as a gzipped TAR archive file. See Export plays, on page 10. |
| 2 | Click to see a pop-up **Play details** window showing the play's description and schema. When you are finished viewing these details, click × or the **Close** button to close the pop-up window. |
| 3 | The **Type** column indicates the type of the play. You can click the column headings (Name, Description, Type, Labels, and Modified by) to sort the table using that column's data. |
| 4 | Click to refresh the plays list. |
| 5 | Click to set filter criteria on one or more columns in the table. |
| | Click to clear any filter criteria you may have set. |

# View the playbook list

The Change Automation application's **Playbook List** window (in the following figure) gives you a consolidated list of all the playbooks in the system. To view the **Playbook List** window, select **Network Automation** > **Playbook List**.

*Figure 2: Playbook List*




| Item | Description |
|---|---|
| 1 | Click + to create a custom playbook. See Create a custom playbook through the UI, on page 12. |
| | Click 🗑 to delete the currently selected custom playbook. See Delete custom playbooks, on page 19. |
| | Click ⬇ to import playbooks from a gzipped TAR archive file. See Import playbooks, on page 19. |
| | Click ⬆ to export the currently selected playbook(s) as a gzipped TAR archive file. See Export playbooks, on page 18. |
| 2 | Click **Manage labels** to assign a label(s) to the playbook. Assigning label(s) to the playbooks allows the system administrator to control which playbooks each user role is allowed to run. |
| 3 | Click ⓘ to see a pop-up **Playbook details** window showing the playbook's description, software compatibility, version number, and its plays. When you are finished viewing these details, click ✕ or the **Close** button to close the pop-up window. |
| 4 | Click the **Name**, **Description**, **Version**, **Software platform**, and **Last modified** column headings in the table to sort the table by that column's data. You can also choose which columns are shown and set quick or advanced filters for any column. |
| 5 | Click ↻ to refresh the playbooks list. |

| Item | Description |
|---|---|
| 6 | Click to set filter criteria on one or more columns in the table. |
| | Click to clear any filter criteria you may have set. |

# About custom plays

Change Automation allows you to create your own custom plays, either based on Cisco models or from scratch. You can also import, export, and delete your custom plays.

You can create custom plays in any of the following types:

- **Check play**: Verifies the data from your devices using a logical expression.
- **Data collection play**: Collects data from your devices.
- **Device config play**: Performs configuration changes on your device
- **Service play**: Provisions and manages a service that is deployed.

**Note** You cannot edit, export, or delete Cisco-supplied plays.

**Note** Check play and Data collection play supports MDT and SNMP collection.

# Create a custom play using templates

This section explains the procedure to create a custom play. The stages of play creation vary depending on the play type you choose:

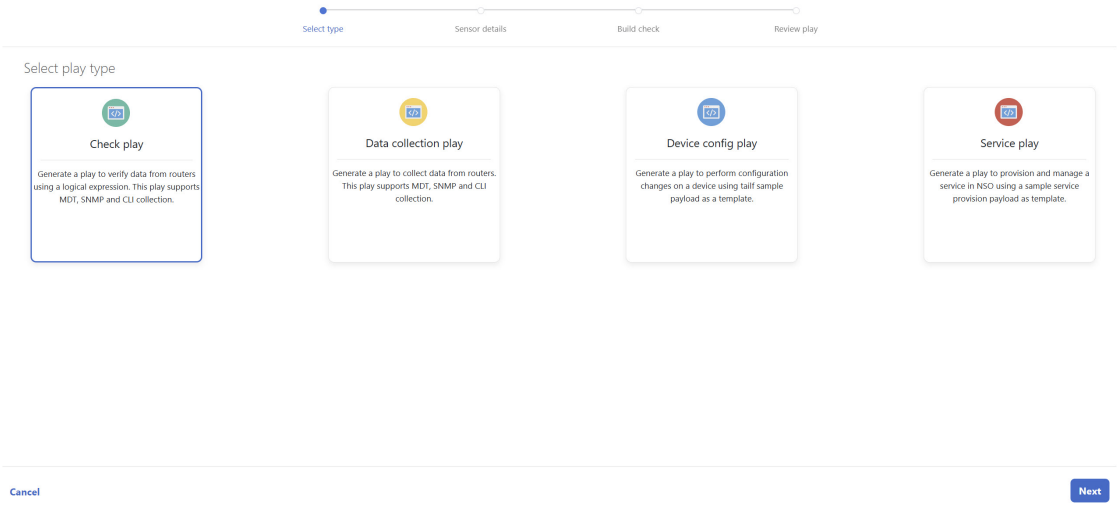
- **Check play**: *Select play type > Select sensor path > Build check expression > Review play*
- **Data collection play**: *Select play type > Select sensor path > Build filter expression > Review play*
- **Device config play** or **Service play**: *Select play type > Configure play (using sample payload in JSON format) > Review play*

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Play List**. The **Play List** window is displayed.

**Step 2** Click to create a custom play. The **Select play type** window opens displaying the types of plays supported and a description for each. The stages of creation are also displayed, and it varies depending on the play type you select.

***Figure 3: Select play type***



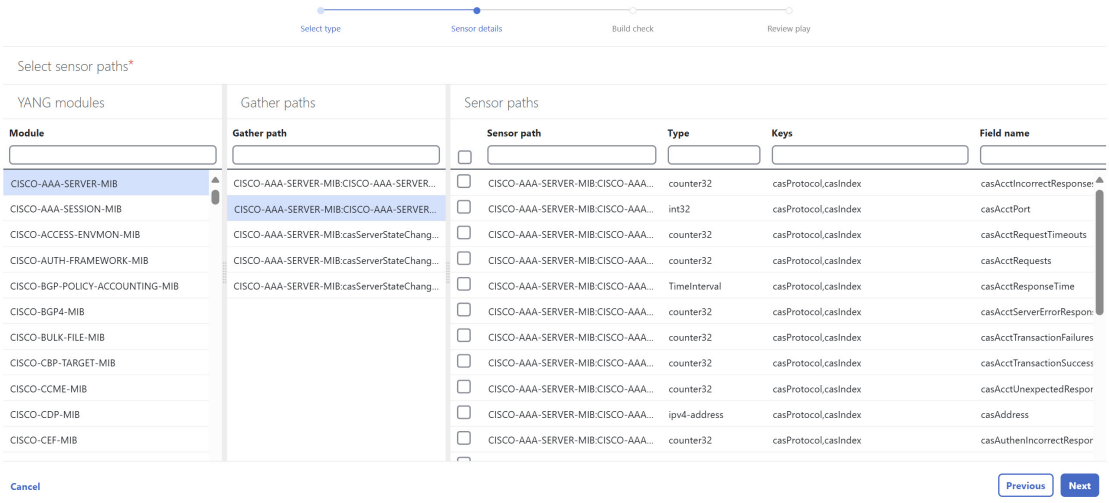

Select the play type that you want to create and click **Next**.

## Step 3 Creating a Check play or Data collection play

When creating Check or Data collection plays, Cisco provides YANG modules for Cisco products. The process that is described in this section assumes that the sensor that you want to use or the field that you want to modify is included in the modules that are provided by Cisco. If the sensor or field is not listed in the default YANG modules, Cisco allows you to expand the device coverage. For information on loading a new or modified module, see Manage device packages in the *Cisco Crosswork Network Controller Administration* guide.

a) In the **Select sensor paths** window, select the required YANG module, Gather path, and Sensor paths. Click **Next** to continue.

***Figure 4: Select sensor paths***

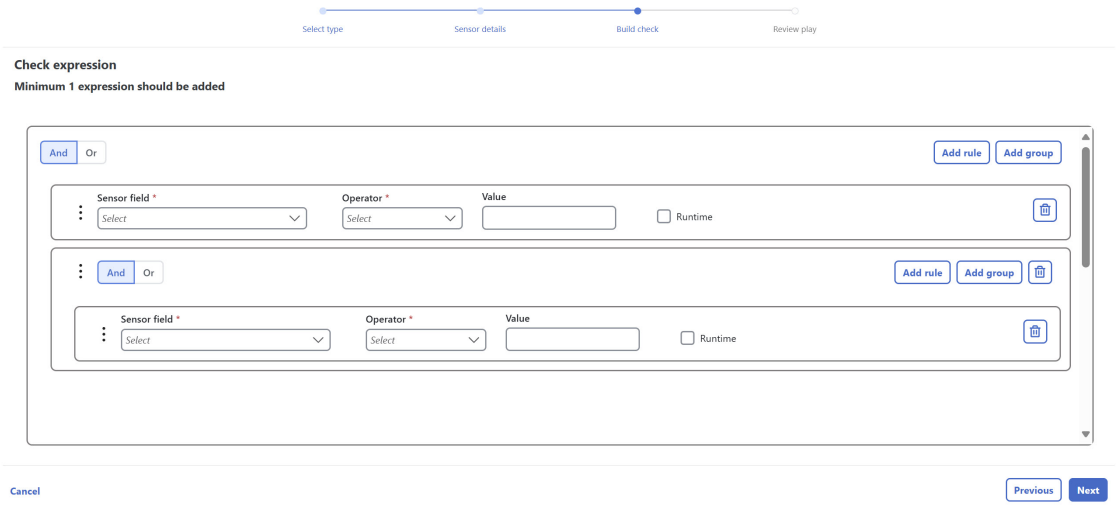



b) Depending on the play type you have selected, you must **Build check** (for Check play) or **Build filter** (for Data Collection play) to apply in your play. Click **Add rule** to add a logic expression using the keys and fields of the

selected sensor path(s). Click **Add group** to add a new logic group. Select the sensor field, operator, and value from the drop-down lists. Select the desired logic operation (AND/OR) between each rule or group.

Click the **Runtime** check box if you prefer to enter the value of the sensor field dynamically during run time. If you select this check box, the *value* field is disabled, and you will be prompted to enter the input parameter when this play is executed (as part of a playbook) during run time.

***Figure 5: Check expression***




Click **Next** to continue.

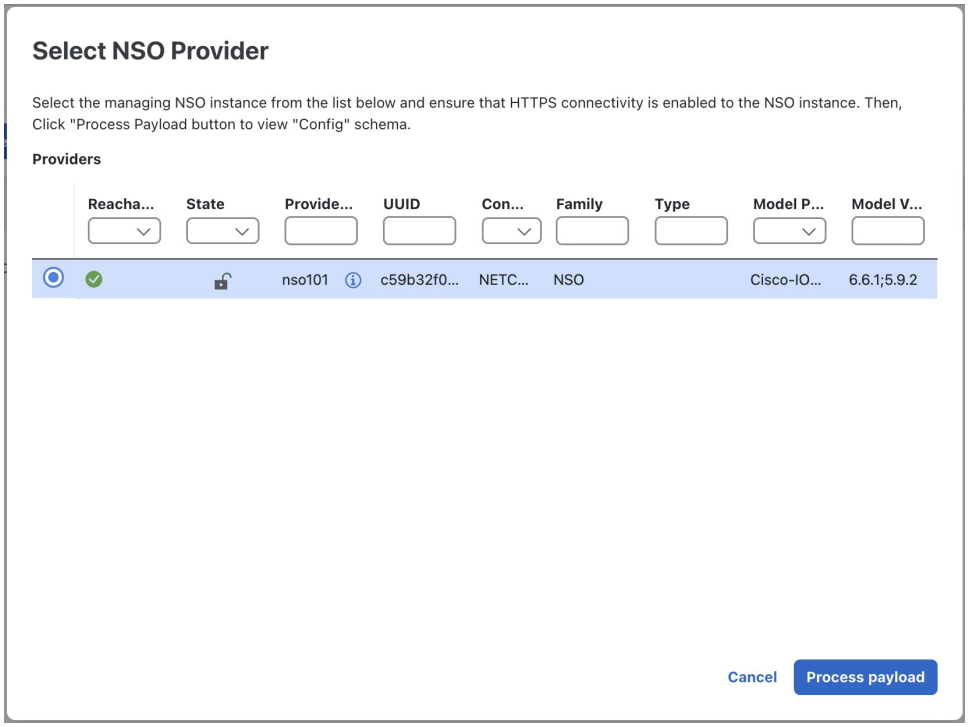
## Step 4 Creating a Device config play or Service play

Ensure that the configuration you are trying to create is available in NSO; otherwise, it will show an error.

When creating a Service play, you are not creating a new service for NSO but creating a play to manage and provision an existing service in one or more NSO instances. For more information, see https://developer.cisco.com/docs/nso/.

a) In the **Configure play** window, click or the **Import** link to import your device config (.JSON) file. You can download and use the sample configuration template. Browse and select your .JSON file, and click **Import**.
b) In the acknowledgment prompt, click **Continue** to select the NSO instance for the config you have imported.
c) Select the NSO provider instance from the dialog box and click **Process Payload**.

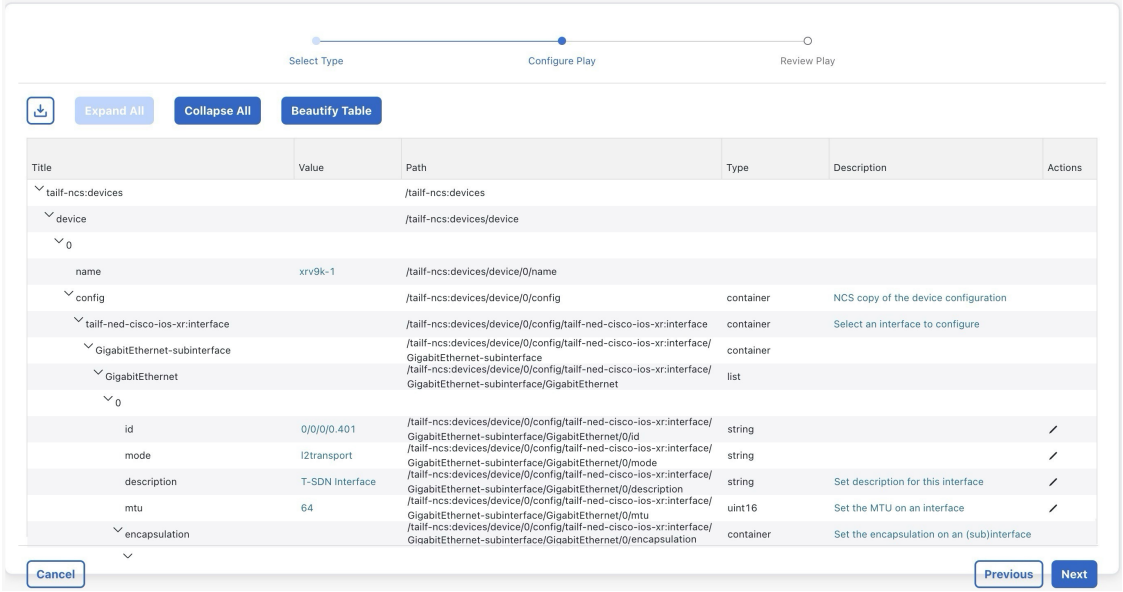
*Figure 6: Select NSO provider*




**Note**
The creation workflow of a Service play is similar to the Device config play, except in the template of the payload file used.

d) The **Configure play** window opens, displaying information from the payload file. You can edit the *value* or *description* columns with the values that you want to see during a playbook execution.

*Figure 7: Configure play*




Click **Next** to continue.

**Step 5** In the **Review play** window, review the parameters of your play. Click **Dry run** to validate your parameters.

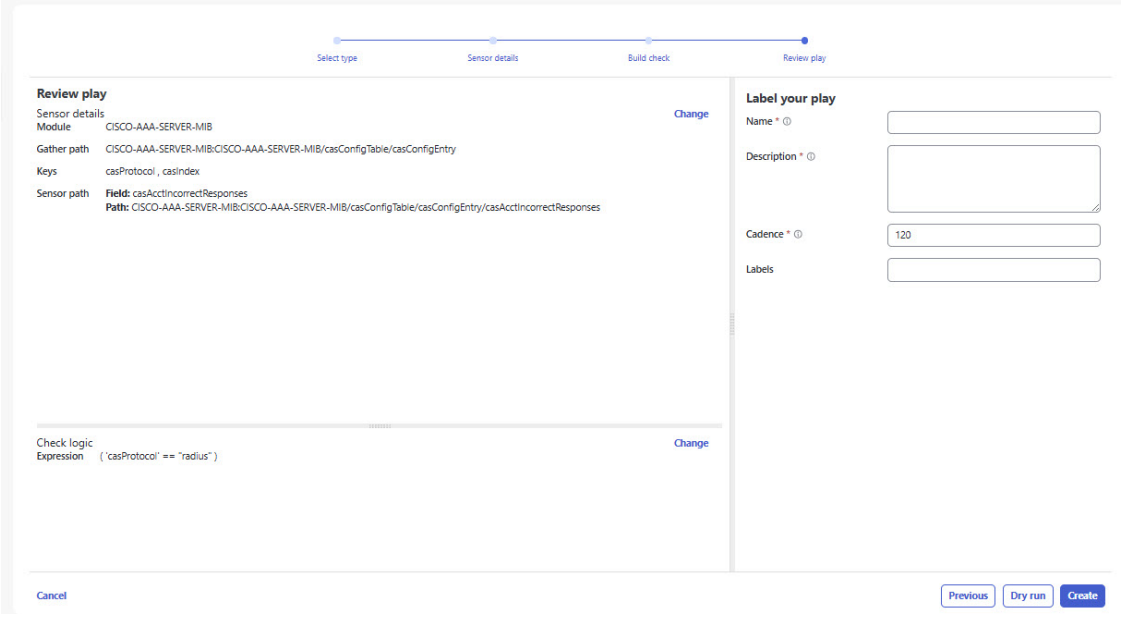
Label your play with a unique **Name** and **Description**.

**Note**
Cisco also formats the play names with indicators such as cfg for configuration, chk for check, and so on, in the name to help you organize the plays properly. You can also use similar tagging for the plays you create.

You can also add labels to your play to group it in the future (optional).

**Note**
The labels determine the type of devices with which you can use the play. For example, an IOS XR play cannot run on IOS XE devices. Be sure to review the labels (IOS XR, IOS XE, and so on) when you add them.

*Figure 8: Review play*




**Step 6** If you are satisfied with your changes, click **Create**.

The **Play List** window opens, displaying your new custom play in the play list.

# Export plays

A user must have Change Automation read permission to export any custom play authored or imported by you or another user into Change Automation.

The exported archive contains only the user-customizable files listed in Playbook components and files, on page 12. Once you extract them from the archive, you can identify the play components by their file names and filename extensions.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Play List**.

**Step 2** Check the check boxes for the custom plays you want to export.

**Step 3** Click . Your browser will prompt you to select a path and the file name when saving the gzipped tar archive. Follow the prompts to save the file.

# Import custom plays

You can import any custom play that meets the following requirements:

- The play files must be packaged as a gzipped tar archive.
- The archive must contain a `.play` file (a data spec file for the play), at minimum.
- The archive file must have a unique name.

**Note** For more details about editing and importing, see Cisco Crosswork Change Automation Developer Guide.

You can overwrite a custom play. The system will warn you when you are about to overwrite a custom play but will not prevent you from doing so.

**Warning** Take precautions to ensure you do not accidentally overwrite the custom plays you created.

### Before you begin

To import plays, a user must have write access. For more information about granting a user read-write role access, see Verify installation and configure system settings.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Play List**.

**Step 2** Click  . Your browser will prompt you to browse to and select the gzipped archive file containing the plays you want to import.

Make sure that there are no Cisco-supplied plays with the same name as the play you intend to import. If you import a play with the same name, it will fail.

**Step 3** Follow the prompts to import the archive file.

# Delete custom plays

You can delete custom plays only. You cannot delete a Cisco-supplied play.

Your user ID must have Change Automation delete permission to delete plays.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Play List**.

**Step 2** In the **Play List** window, select the custom plays you want to delete.

**Step 3** Click the  icon.

**Step 4** When prompted, click **Delete** again to confirm.

# About customizing playbooks

You can create your own playbooks from scratch, based on details from Cisco-supplied playbooks. You can also create custom playbooks using the available plays.

Creating and modifying Cisco-supplied playbooks are engineering tasks that take place outside of the user interface for Change Automation. As such, they are outside the scope of this User Guide.

Cisco supplies developer-level documentation for Cisco-supplied playbooks. For more information on how to create custom plays and playbooks, see Developer Guides on Cisco DevNet.

## Playbook components and files

Change Automation playbooks contain various components, referred to using specialized names. The components are implemented in the playbook as files. Some of these components' names are borrowed from the Ansible specification, but all have their definitions, and not all the corresponding files can be customized by users. Some components are Cisco proprietary intellectual property; while you can use them in custom plays and playbooks, you cannot customize them directly. For more information, see Writing custom playbooks at Cisco Crosswork Change Automation developer guide.

## Create a custom playbook through the UI

Change Automation allows users with admin and read/write roles to create custom playbooks using the available plays. For more information about granting read/write role access to a user, see the section "Assign Change Automation User Access Levels" in the topic Verify installation and configure system settings.

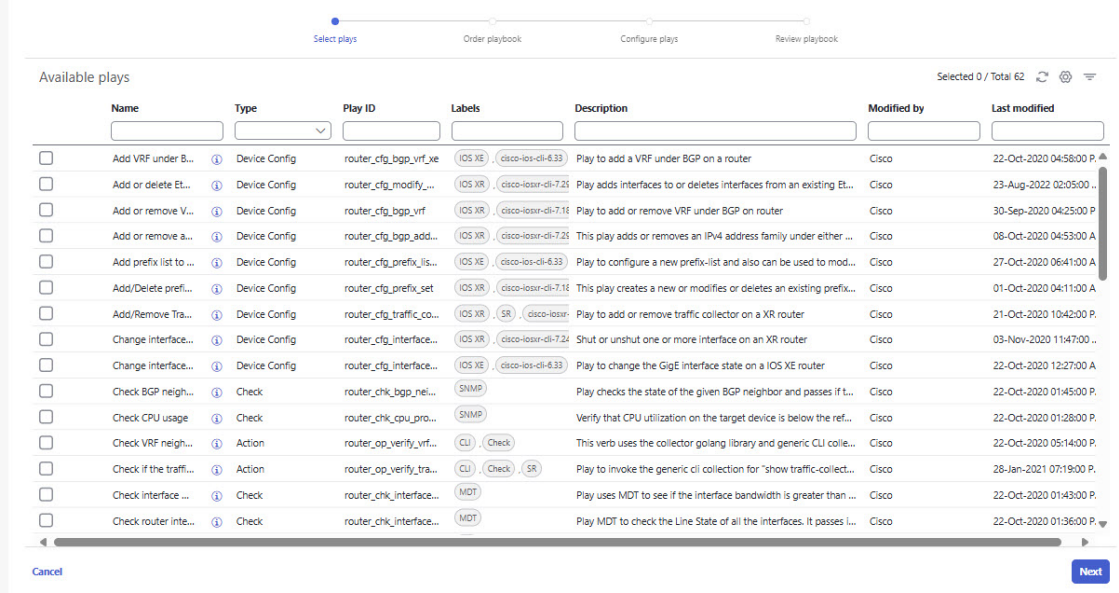
**Note** You cannot edit a custom playbook once it is created. We recommend that you perform a dry run of the playbook *before completing the creation* to ensure that the playbook's purpose is met. Once created, if you must make changes to the custom playbook, you have to recreate your playbook with the relevant changes.

**Procedure**

**Step 1** From the main menu, choose **Network Automation** > **Playbook List**. The **Playbook List** window is displayed.

**Step 2** Click  to create a custom playbook. The **Select plays** window opens displaying the available plays.

*Figure 9: Select plays*




Select all the plays you want in your playbook, and click **Next**.

**Note**
The recommendation is to include the **Perform Check Sync on the device** and **Sync NSO from device** plays as a pre-step to running other operations in the playbook or as part of pre-maintenance.

The **Perform Check Sync on the device** play checks the device sync status with NSO and performs sync only when needed, based on the playbook's sync parameter value. It reduces the playbook execution time and ensures the NSO configuration matches the device configuration.

- If the playbook's sync parameter is set to True and the device is not in sync, the **Perform Check Sync on the device** play will sync the device with the NSO configuration.
- If the sync parameter is set to False and the device is not in sync, the play will fail to execute with a commit message.
- If the device is already in sync, the play will succeed.

**Step 3** In the **Order playbook** window, arrange the order of the plays in the playbook as per the execution phase (Continuous, Pre-Maintenance, Maintenance, Post-Maintenance). By default, all the selected plays are displayed within the Maintenance phase. You can click and drag the plays to rearrange them to the appropriate phase.

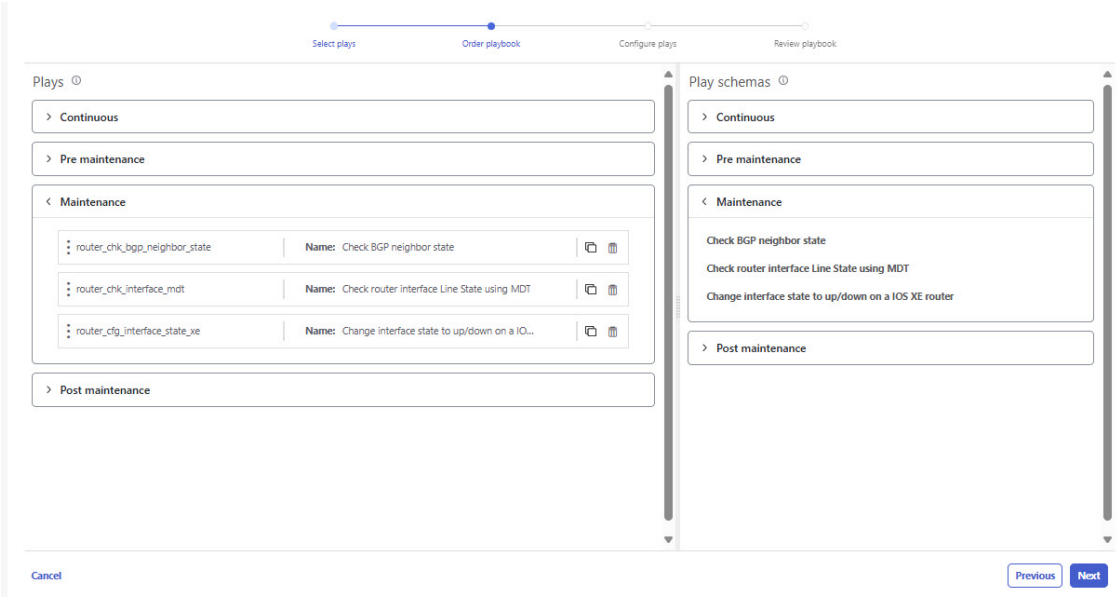
Depending on the type of play you have selected, it may be restricted from being used in certain phases. For example, a configuration play cannot be used outside of the Maintenance phase.

For more information on each execution phase, see Playbook execution order, on page 21.

Change Automation also formats the play names with indicators such as "cfg" for configuration, "chk" for check, and so on, in the name to help you organize the plays properly. You can use similar tagging for the plays you create.

You can also duplicate or delete a play by clicking on the icons provided.

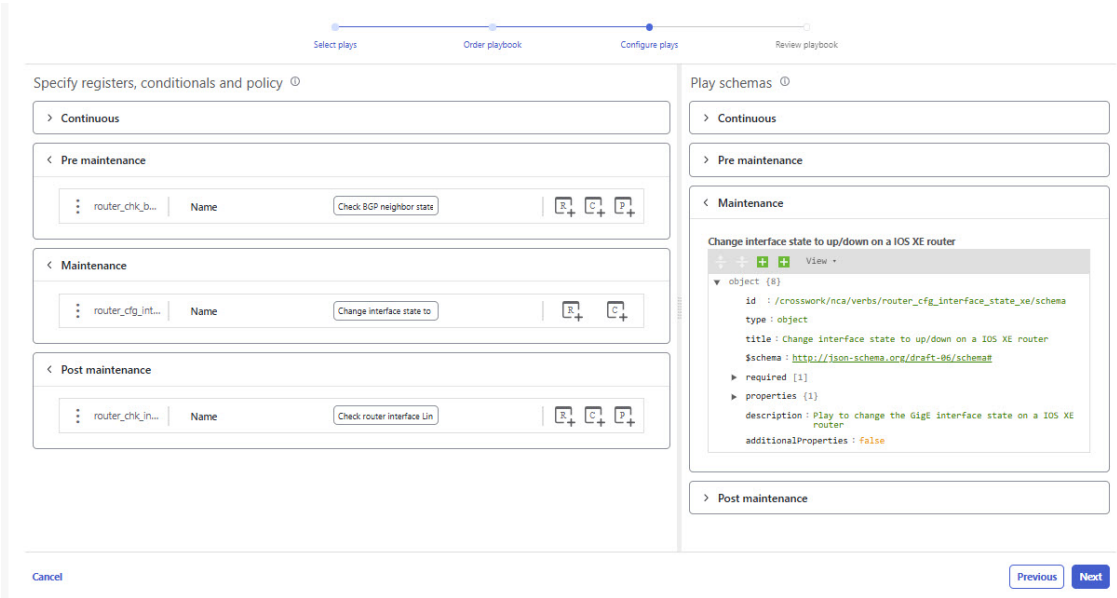
*Figure 10: Order playbook*




Click **Next**.

**Step 4** The **Configure plays** window opens, displaying the plays in each execution phase and the play schemas.

*Figure 11: Configure plays*



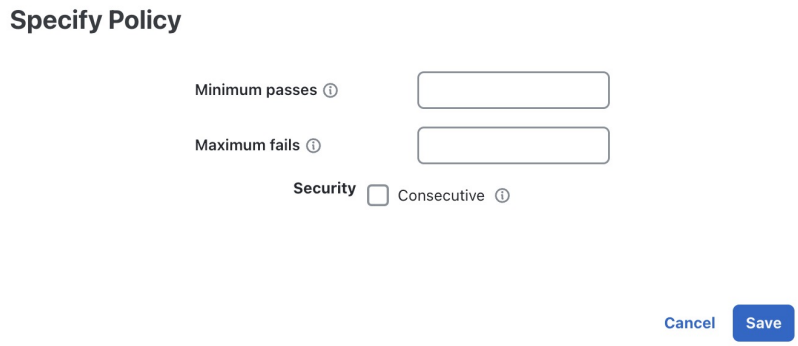

You can perform the following:

- Click to specify a policy for a play. In the **Specify policy** dialog box, specify relevant values for the fields provided. Click for more information about each field. Click **Save** to save your policy values.

  **Note**

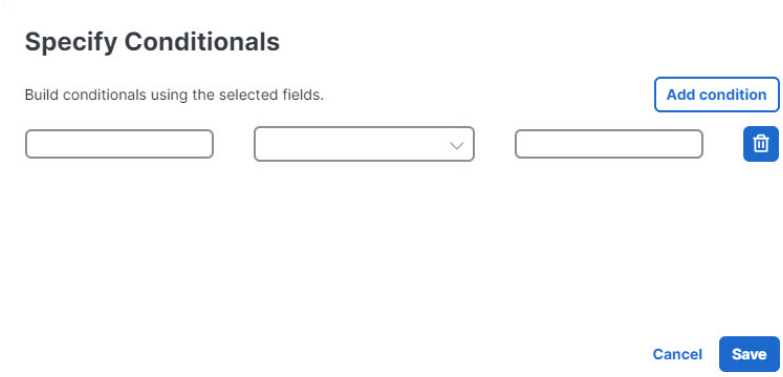
Policies are applicable to Check plays.

***Figure 12: Specify policy***



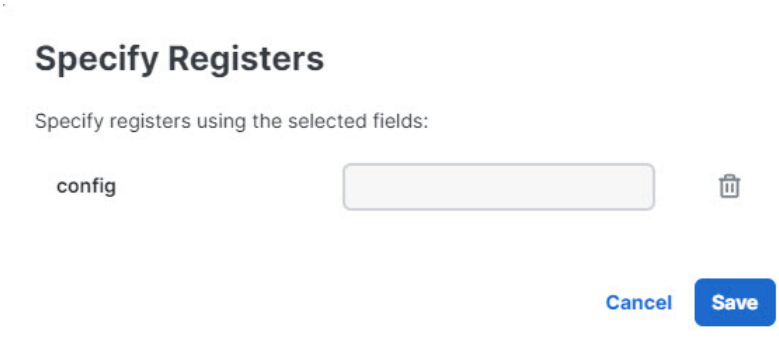

- Click to apply a condition to a play. Execution of the play proceeds only if the condition is met. In the **Specify Conditionals** dialog box, click **Add Condition** to add a condition. Click **Save** to save your conditional values.

***Figure 13: Specify conditionals***




- Click to specify a register for a play. Specifying registers allows you to use the output of a previous play as the input for another play. Click **Save** to save your registers.

***Figure 14: Specify registers***

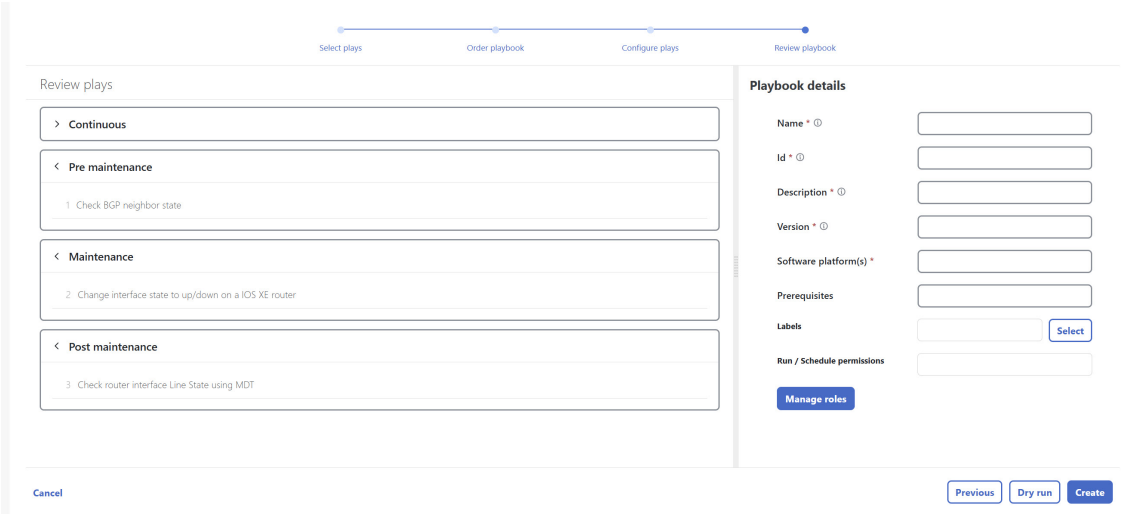



- (Optional) Rename the plays if you want them to be displayed with different names during the playbook execution.

Click **Next** to continue.

**Step 5** In the **Review playbook** window, review the plays in your playbook. Enter relevant values for the **Playbook details** fields. You can click ⓘ for more information about each field.

***Figure 15: Review playbook***




**Note**
For the **Software platform(s)** field, make sure to use the exact software type name as it is mentioned in **Device Management** > **Network Devices** > **Software type** column.

**Step 6** (Optional) Click **Select** and perform one of the following, as applicable, to set the **Labels**:

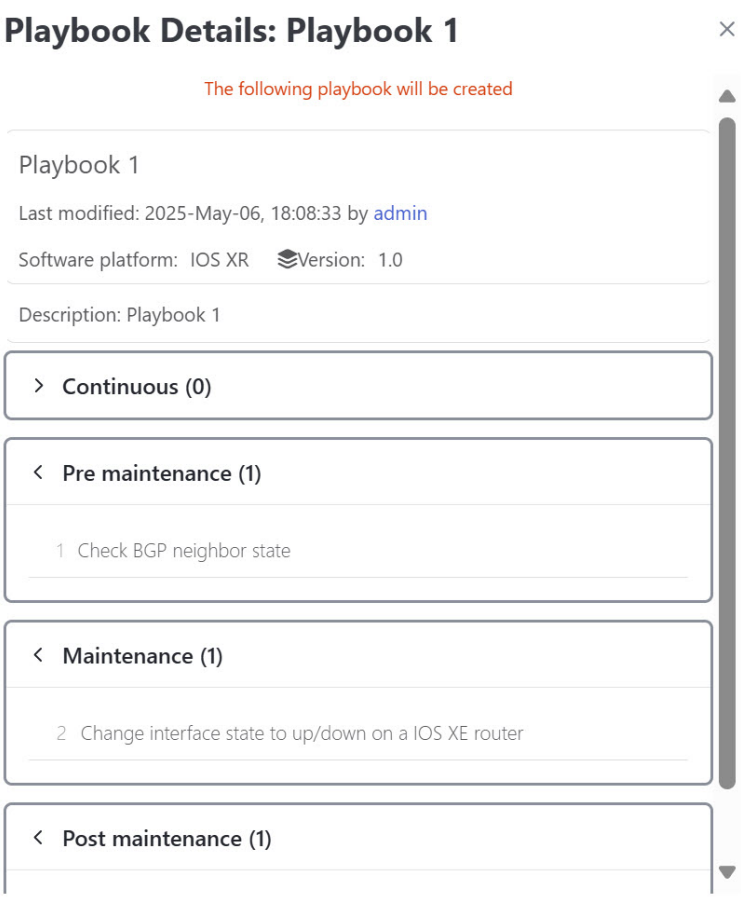
- Select the applicable label and click **Done**.
- Click **+ New label**, enter relevant values for **Label** and **Roles**, and click **Save**. Select the new label and click **Done**.

**Note**
The labels determine which user or roles can run which playbooks.

For more information on assigning playbooks to specific roles, see Assign playbooks to specific roles, on page 20.

**Step 7** (Optional, but recommended for testing the playbook) After you enter the relevant details, click **Dry run** to validate the parameters. A dialog box opens, displaying the playbook Details.

*Figure 16: Playbook details*




**Note**
Dry run does not commit the changes but provides a platform to validate whether the playbook would work with the parameters you entered.

**Step 8** Click **Previous** to navigate back to a step to make changes as necessary to get the playbook to function properly.

**Step 9** Click **Create** to create the playbook.

The **Playbook list** window opens, displaying your new custom playbook on the list.

# Create a custom playbook using APIs

This section explains the steps to create a custom playbook using APIs. For more information, see the Writing custom playbooks tutorial on Cisco DevNet.

**Note** A playbook containing a custom play can be created through the UI (see Create a custom playbook through the UI, on page 12) or using APIs.

A playbook consisting of one or more custom plays is expected to have a *dataspec* value for the custom play in the playbook file. The *dataspec* value is generated when the custom playbook is created using the API in this procedure. You cannot create the same custom playbook using the import option (API: **/v1/mops/import**), as it does not add the *dataspec* value for the custom play.

### Procedure

**Step 1** Ensure that the plays (stock or custom) you need for the playbook are created beforehand.

You can create a custom play either through the UI (see Create a custom play using templates, on page 5) or using API (use the API call **//crosswork_ip:30603/crosswork/nca/v1/Plays/device/config**).

**Note**
If you are importing plays that share the same name with existing plays, then the error "`Play validation failed, custom Play already present`" will be displayed, to prevent the existing plays being overwritten.

**Step 2** Create the playbook using the following API:

API call: **//crosswork_ip:30603/crosswork/nca/v1/mops**

# Export playbooks

You can export any playbook as a gzipped tar archive. This includes any Cisco-supplied playbook and custom playbooks you or another party have authored and imported into Change Automation.

The exported archive contains only the user-customizable files listed in Playbook components and files, on page 12. It also contains one or more `.pb` files (for example, `router_config_bgp_rd.pb` for the playbook code), which are parsed and processed at the back end.

You can edit the exported files as needed, following the guidelines in the Writing custom playbooks tutorial on Cisco DevNet. Then, you can import them as explained in Import playbooks, on page 19.

Your user ID must have Change Automation read permission to export playbooks and write permissions to import new or modified playbooks.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Playbook List**.

**Step 2** (Optional): In the **Playbook List** window, filter the table as needed.

**Step 3** Check the check boxes for the playbooks you want to export. Check the check box at the top of the column to select all playbooks for export.

**Step 4** Click . Your browser will prompt you to select a path and the file name when saving the gzipped tar archive. Follow the prompts to save the file.

# Import playbooks

You can import any custom playbook, provided it meets the following requirements:

- The playbook files must be packaged as a gzipped tar archive.
- The archive must contain a `.pb` file, at minimum.
- The archive file must have a unique name.

The individual files included in the archive must meet the additional validation requirements described in the Writing custom playbooks tutorial on Cisco DevNet.

**Note** While you cannot overwrite a Cisco-supplied playbook, you *can* overwrite a custom playbook. The system will warn you when you are about to overwrite a custom playbook but will not prevent you from doing so. Take precautions to ensure that you do not overwrite your custom playbooks accidentally.

You cannot re-import an exported Cisco-supplied playbook with the same name as the original.

### Before you begin

To import playbooks, a user must have write access. For more information about granting a user read-writer role access, see Verify installation and configure system settings.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Playbook List**.

**Step 2** Click . Your browser will prompt you to browse to and select the gzipped archive file containing the playbooks you want to import.

Make sure there is no existing playbook with the same name as the playbook you intend to import unless you want to overwrite the existing playbook.

If you are creating an improved version of a playbook, it is recommended that you use a version number or other indicator to ensure that the name is unique and does not overwrite the original playbook until the replacement is completely tested.

**Step 3** Follow the prompts to import the archive file.

# Delete custom playbooks

You can delete user-defined playbooks only. You cannot delete a Cisco-supplied playbook.

Your user ID must have Change Automation delete permission to delete playbooks.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Playbooks List**.

**Step 2** In the **Playbooks List** window, select the custom playbook you want to delete.

**Step 3** Click the icon.

**Step 4** When prompted, click **Delete** again to confirm.

# Assign playbooks to specific roles

This section explains how to assign playbook labels to specific roles so that they can run and import the playbooks with that particular label. Admin users can enable other users to run playbooks with a specific label.

### Before you begin

If required, create a new user to whom you would want to assign the playbook. For more information, see the topic Create user roles in the *Cisco Crosswork Network Controller Administration* guide.

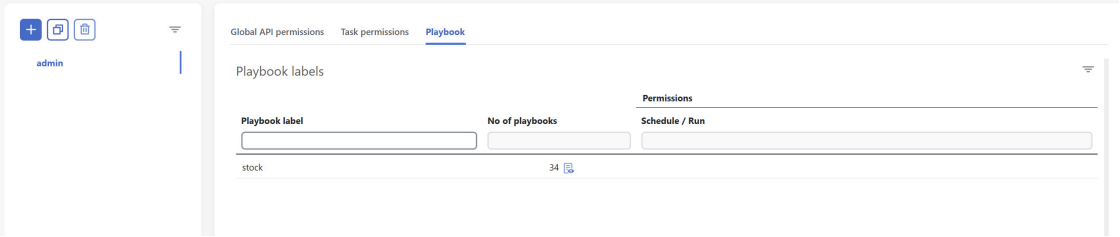
### Procedure

**Step 1** Go to **Administration** > **Users and Roles** > **Roles** > **Playbook**.

**Step 2** Under the **Roles** pane, select the role to whom you want to assign the playbook labels.

**Step 3** Enable the **Permissions** check boxes for the **Playbook label**(s) you want to assign.

***Figure 17: Playbook labels***




# About running playbooks

You must have permission to run the playbooks with a particular playbook label. For more information on assigning playbooks to specific roles, see Assign playbooks to specific roles, on page 20.

Running any playbook consists of five steps:

1. Select the **playbook** you want to run (see View the playbook list, on page 3).

2. Select the **device or devices** that you want to run it on.
3. Enter the appropriate run-time **parameters** that you want the playbook to apply.
4. Select the **execution mode** that you want to use:
   a. Perform a dry run of a playbook, on page 22, where you can see what the playbook does before make changes to the network.
   b. Run playbooks in single stepping mode, on page 28, so you can pause after each playbook check or action, and roll back changes you did not intend.
   c. Run playbooks in continuous mode, on page 34 and apply the changes immediately.

   While selecting the execution mode, you can also choose to:
   - Schedule playbook runs, on page 40 for another calendar date or time.
   - **Collect syslogs** during and after the run. Syslog collection is available only when running the playbook in single-stepping or continuous execution mode and only if you have already configured a syslog storage provider.
   - Specify a **Failure policy**, where you decide what the system should do if a failure occurs during the playbook run.
5. **Confirm** your settings and run the playbook in the execution mode you selected.

Depending on their complexity and network factors, some playbooks may take much time to run. You can view the run details and status at any time during and after the completion of a run. If the playbook is still running, you can also choose to cancel it. For details, see View or abort playbook jobs, on page 42.

# Playbook execution order

When it is running, every playbook conducts checks and configuration changes in four phases, which correspond to sections of the playbook code (identified using the tags discussed in Playbook components and files, on page 12):

1. **Pre-Maintenance**—This phase of the playbook includes non-disruptive checks and any other operations on the device that prepare it for potentially traffic-impacting changes. For example:
   - Take snapshots of various routing protocol states.
   - Take snapshots of memory, CPU, and system health parameters.
   - Validate the capacity (storage, memory) on active and standby routers for the new software patch upgrade.
2. **Maintenance**—This phase of the playbook includes any task that may disrupt traffic flowing through the router or impact neighboring routers. For example:
   - Cost out the router and wait until traffic drains out completely.
   - Verify that the redundant router is healthy and carrying traffic.
   - Perform the upgrade procedure on the device.
   - Reconfigure the device(s) to support a new configuration or feature.

3. **Post-Maintenance**—This phase of the playbook includes verification tasks to perform on the router after any disruptive operation. For example:
   - Verify that the current state matches the desired state.
   - Cost in the router and wait for traffic to return to normal levels.
4. **Continuous**—In addition to the three serial phases already described, Change Automation can also run check tasks that span the entire duration of playbook execution. These tasks check the state of the router while the playbook is being deployed and cancel the playbook execution if any catastrophic or undesirable state change occurs. The checks in the playbook may also monitor a neighboring router to guarantee no second-order failures in the network while the changes are being deployed.

# Perform a dry run of a playbook

A dry run lets you view configuration changes that the playbook will send to the device without performing the actual commit of the changes, as you would with a run in the single-stepping or continuous execution modes.

It is a best practice to perform a dry run and verify the configuration changes before you deploy those changes to the router. If the dry run fails, you may want to debug its parameter values using another dry run. You can also debug by performing a single-stepping run, which will allow you to abort and rollback changes after one or more of the plays, instead of only at the end, as part of a continuous run's Failure Policy.

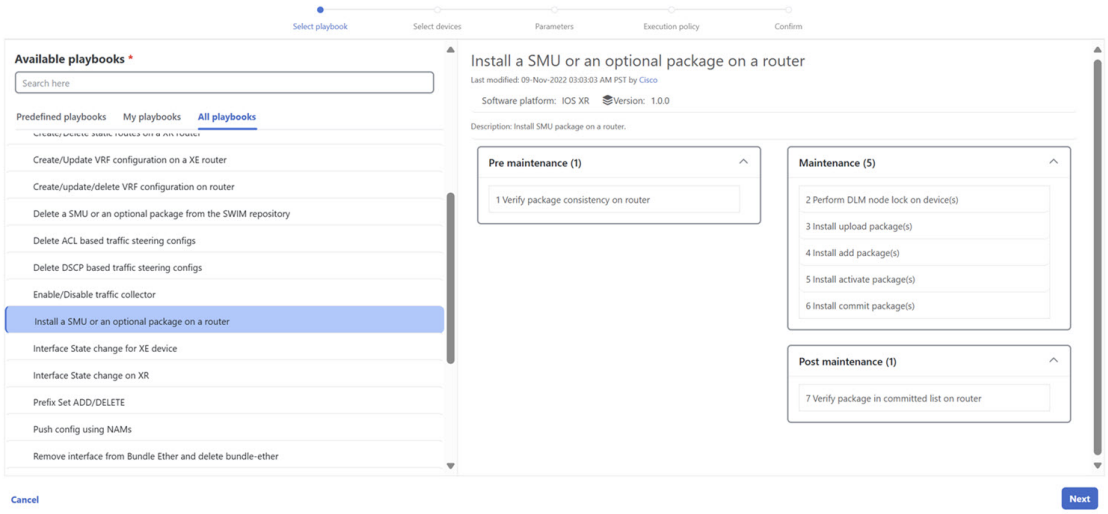
Note that dry run mode is intended for use only with playbooks that perform device configuration changes via Cisco NSO. Some playbooks do not support dry run mode. For example, **Install a SMU or an optional package on a router** and **Uninstall an optional package or a SMU**.

### Procedure

**Step 1** From the main menu, choose **Network Automation** > **Run Playbook**.

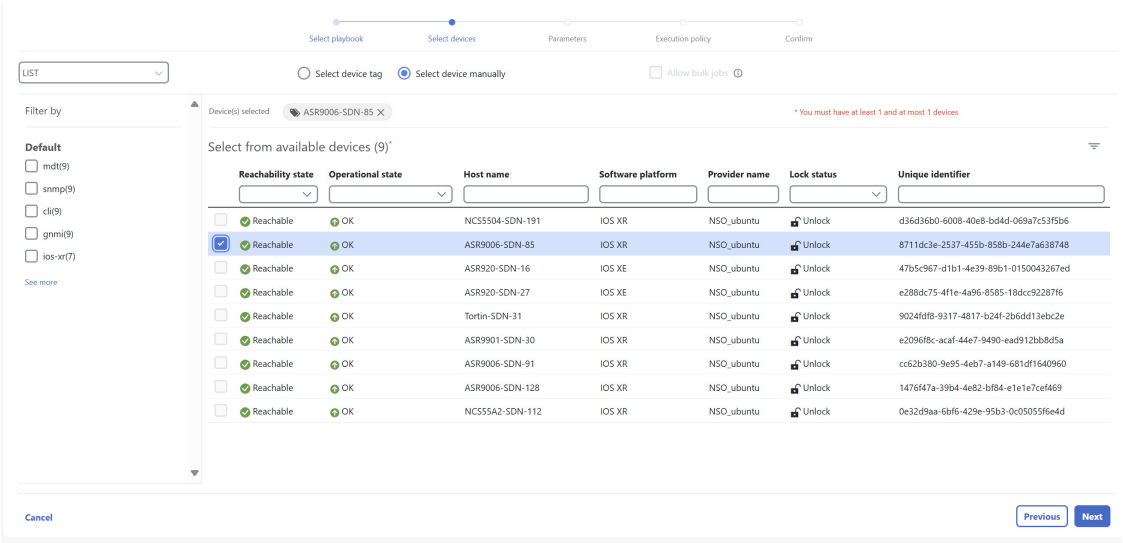
**Step 2** In the **Available playbooks** list on the left, click on the playbook you want to dry run. On the right, the window displays the playbook name, hardware and software compatibility information, and descriptions for all the plays in the selected playbook.

*Figure 18: Select playbook*




**Step 3** Click **Next**. The **Select devices** window appears.

*Figure 19: Select devices*




Using this window:

- You can toggle between the table view and topology map view by clicking and selecting the relevant option on the drop-down button on the upper left corner of the window. Choose **Select devices from list** or **Select devices from map** to select the table view or topology map view respectively. By default, the table view is displayed.
- With the topology map view displayed, you can toggle between the map's geographical and logical views by clicking on the or the . You can also zoom, display bandwidth utilization, and change logical view layouts as you do with the topology map you see when you select **Topology** from the main menu.
- You can select the devices manually or using tags. The **Select device tag** option targets you to select the relevant tag instead of devices from the table and all devices associated with the relevant tag are selected. The **Select device**

**manually** option allows you to select the devices from the list using quick and advanced filters and filter by tags on the left. Hover the mouse pointer over the ⓘ icon next to the options for more information. You can also view the selection criteria, such as the number of devices required for the selected playbook.

**Note**
If you are a non-admin user and selecting the devices manually, make a note of the following:

- The devices on which you want to run the playbook must belong to a Device Access Group, and you must have access to this Device Access Group. For more information on Device Access Groups and associating a user with a Device Access Group, see the Manage Device Access Groups section in the *Cisco Crosswork Network Controller Administration* guide.
- If your role is associated with an empty Device Access Group, then you will receive an error message.
- If your role is associated with multiple Device Access Groups and the device belongs to any of these Device Access Groups, then you can run the playbook on this device. If the device does not belong to any of your Device Access Groups, then the operation fails.
- If you are selecting multiple devices (using **Allow bulk jobs** option or using tags) and if any of the devices does not have access, then an error message appears stating that this list of devices does not have access to run the playbook.

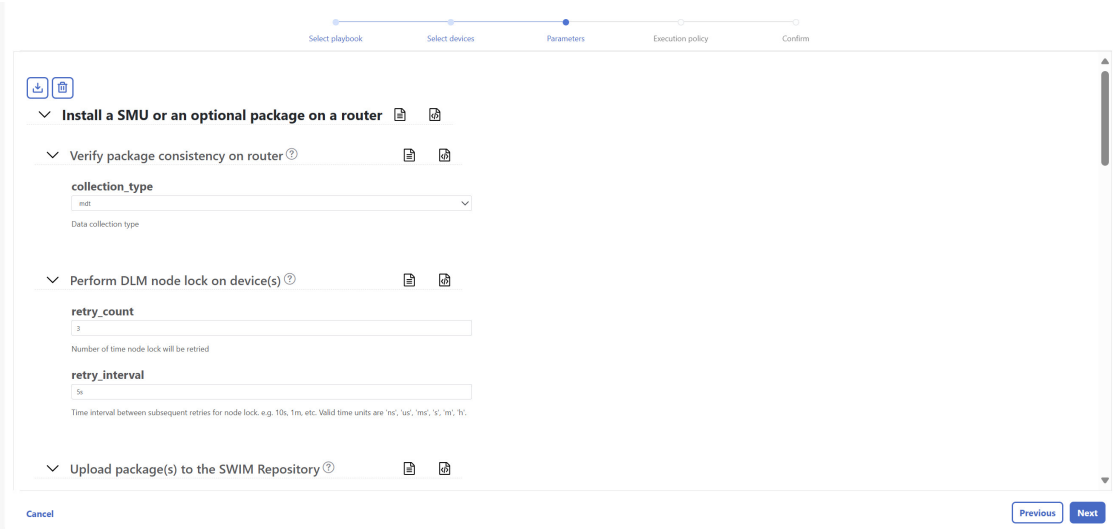
- In the **Select device manually** selection mode, you can check the **Allow bulk jobs** check box to select multiple devices and run the selected playbook on them simultaneously. Based on your selection, the system creates a static group of multiple jobs. Hover the mouse pointer over the ⓘ icon next to the check box for more information. There is no limit to the number of devices you can select for a bulk job.

**Note**
**Allow bulk jobs** option is enabled for playbooks that can be executed on a single device.

**Step 4** Click **Next**. The **Parameters** window appears.

**Step 5** In the fields provided in the **Parameters** window, enter the playbook parameter values for this dry run.

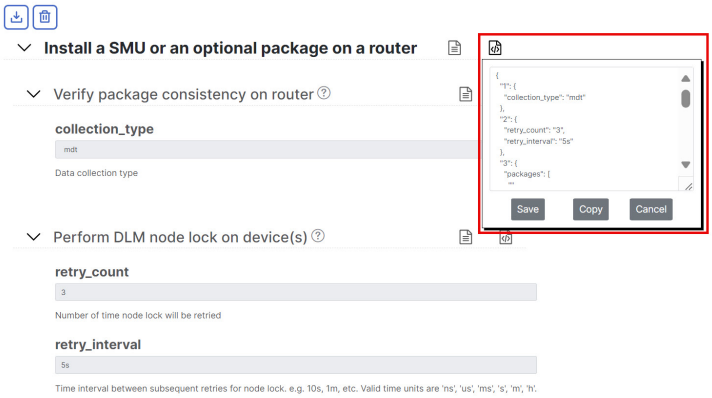
***Figure 20: Parameters***



With the **Parameters** window displayed, you can also:

- Click to upload a JSON file with the parameter values you want. You will be prompted to navigate to the JSON parameters file you have previously prepared (or downloaded from a previous playbook run) and then upload it as appropriate for your browser and operating system.
- Click to enter the parameter values in JSON format. A popup text window displays the full list of JSON parameters with empty values in quotes. Edit the values, and when you are finished, click **Save**.

***Figure 21: Edit JSON***




- Click to select the items you want to configure. A pop-up text window will appear, displaying the complete list of plays or parameters that can be configured. For the playbook, it will show a list of plays and, for each selected play, it will display a list of parameters that can be configured for that play. If you unselect any item, it will not appear for configuration. Uneditable options indicate mandatory items that cannot be deselected.

***Figure 22: Object properties for a playbook***

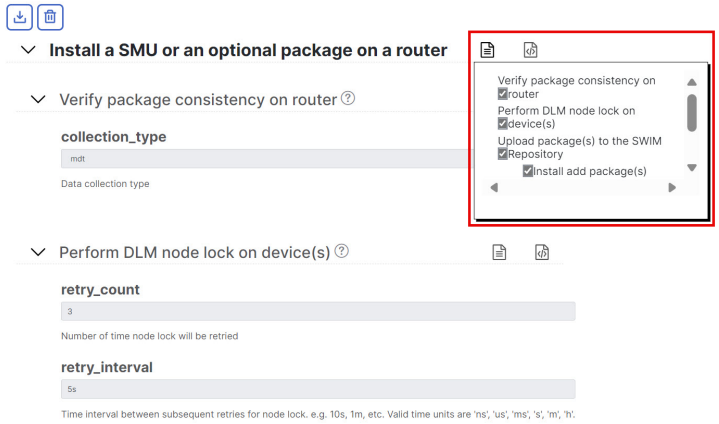
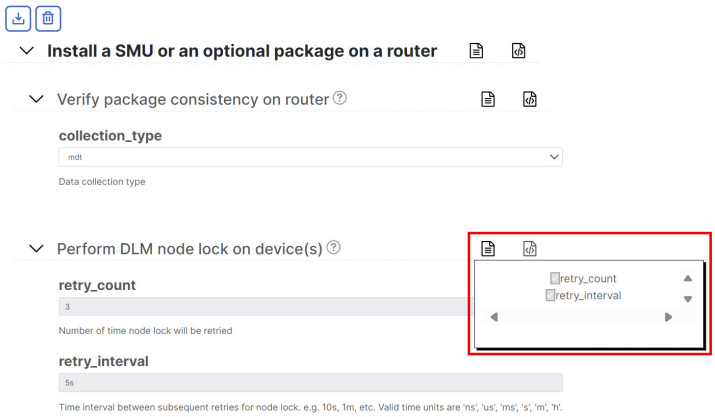


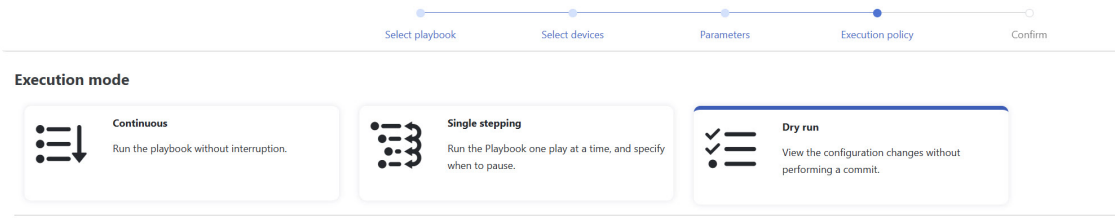
*Figure 23: Object properties for a play*




- Click ⊕ to add additional instances of a particular parameter, if required for the playbook you are running. Click 🗑 to delete instances added in this way.
- Click 🗑 to clear all the parameter values entered so far.

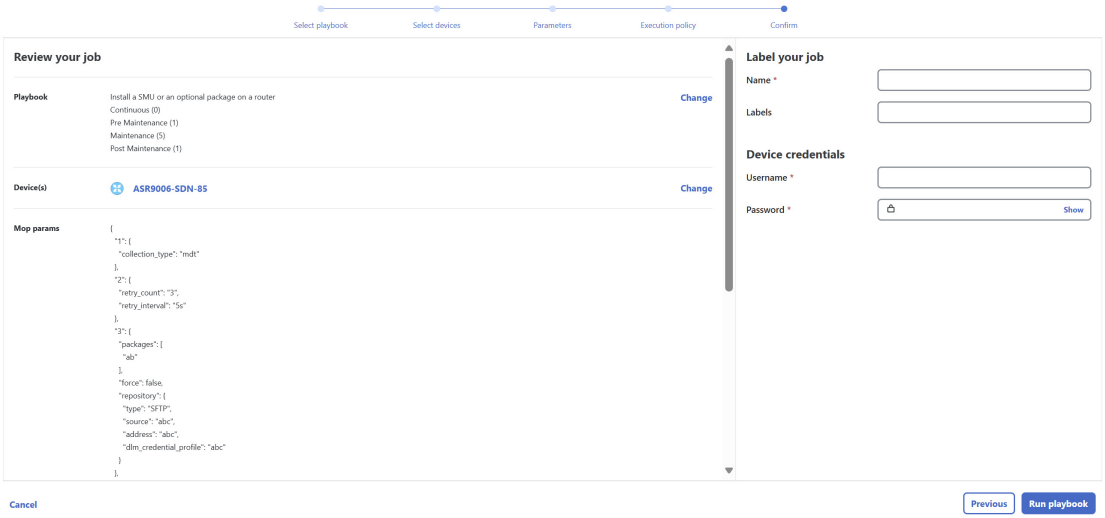
**Step 6** With the parameter values set, click **Next**. The **Execution policy** window appears.

*Figure 24: Execution Policy*




**Step 7** Choose **Dry run** and click **Next**. The **Review your job** window appears, displaying a summary of your choices: playbook, devices, parameters, and execution policy.

*Figure 25: Review your Job*




In this window:

- You must provide a relevant **Name** for the job.
- You can enter labels for your job using the **Labels** field.
- You can click on any **Change** links in the **Review your job** window summary to modify your choices.

**Step 8** (Optional) Enter the device credentials (name and password).

**Note**
This step is applicable only if **Credential prompt** is enabled in the Change Automation settings. For more information, see Verify installation and configure system settings.

**Step 9** When you are ready to continue, click **Run playbook**.

**Step 10** At the confirmation prompt, click **Confirm**. The **Execution mode** window is displayed.

**Step 11** After the dry run is complete:

- Click the **Dry run** tab and verify the configuration changes that would be pushed to the device had this not been a dry run. This tab will display a `no config change` message if no changes have been made. Please note that this tab shows only cumulative configuration changes, not each individual change made. For example, if a playbook configures `set-overload-bit` in one step and then unconfigures it using `no set-overload-bit` later, the tab will show `no config change`.
- Click the **Events** tab to see success and failure messages for each step of the playbook. This can help you diagnose and correct problems with individual plays and the run as a whole. For troubleshooting information, see Troubleshoot change automation, on page 45.
- Click the **Console** tab to see messages that are generated during the run.

As syslog collection is disabled for dry run, the **Syslog** tab will contain only a message stating that.

**Step 12** (Optional) If you want to perform a single-step debugging run, or are ready to commit the changes to the device, click **Execute now**. The **Execution policy** window will display all of your parameter values from the dry run.

# Run playbooks in single stepping mode

Single-stepping execution mode is a handy way to test a custom or modified playbook or diagnose problems with a pre-packaged playbook that does not give you the desired results. Unlike a dry run, a single-stepping execution commits configuration changes to the device as the playbook runs. However, you can set breakpoints on or pauses after any Maintenance or Post-Maintenance action in the playbook. Note that while you can set breakpoints on Pre-Maintenance actions, doing so will have no effect, and these actions will not pause.

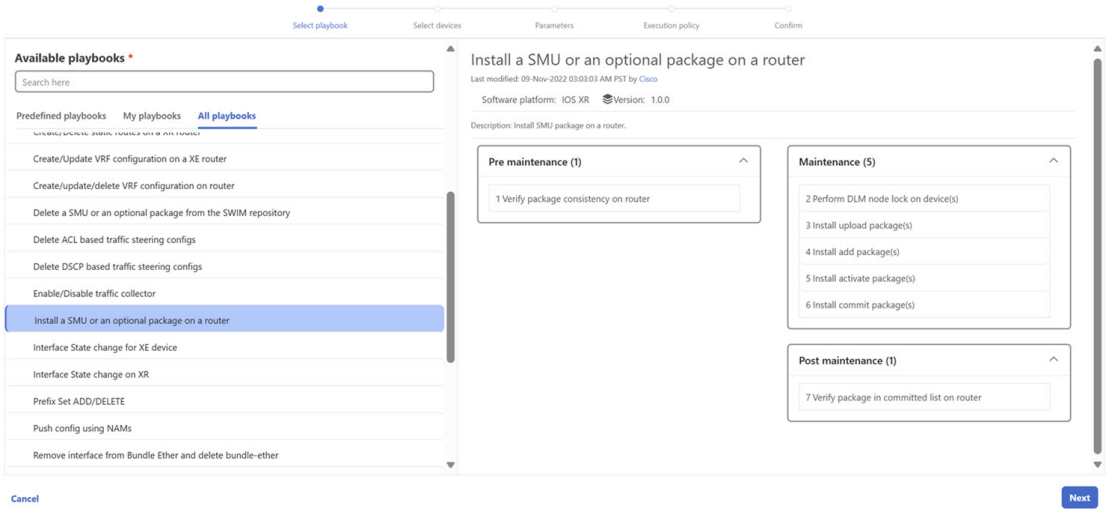
Whenever the playbook hits a breakpoint, it will stop and will not continue until you issue the command to proceed. At each pause, you can also abort the entire run and roll back all changes made or roll back to any previous play.

## Procedure

**Step 1** From the main menu, choose **Network Automation** > **Run Playbook**.

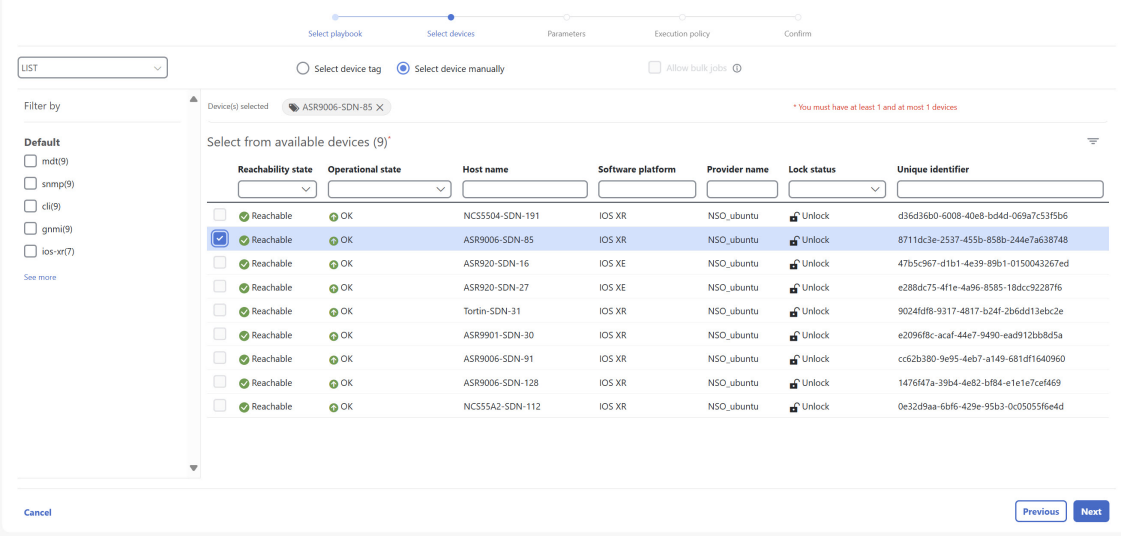
**Step 2** In the **Available playbooks** list on the left, click on the playbook you want to run. On the right, the window displays the playbook name, hardware and software compatibility information, and descriptions for all the plays in the selected playbook.

***Figure 26: Select playbook***




**Step 3** Click **Next**. The **Select devices** window appears.

*Figure 27: Select devices*




Using this window:

- You can toggle between the table view and topology map view by clicking and selecting the relevant option on the drop-down button in the upper left corner of the window. Choose **Select devices from list** or **Select devices from map** to select the table view or topology map view, respectively. By default, the table view is displayed.
- With the topology map view displayed, you can toggle between the map's geographical and logical views by clicking on the or the . You can also zoom, display bandwidth utilization, and change logical view layouts as you do with the topology map you see when you select **Topology** from the main menu.
- You can select the devices manually or using tags. The **Select device tag** option targets you to select the relevant tag instead of devices from the table and all devices associated with the relevant tag are selected. The **Select device manually** option allows you to select the devices from the list using quick and advanced filters and filter by tags on the left. Hover the mouse pointer over the icon next to the options for more information. You can also view the selection criteria, such as the number of devices required for the selected playbook.

  **Note**
  If you are a non-admin user and selecting the devices manually, make a note of the following:

  - The devices on which you want to run the playbook must belong to a Device Access Group, and you must have access to this Device Access Group. For more information on Device Access Groups and associating a user with a Device Access Group, see the Manage Device Access Groups section in the *Cisco Crosswork Network Controller Administration* guide.
  - If your role is associated with an empty Device Access Group, you will receive an error message.
  - If your role is associated with multiple Device Access Groups and the device belongs to any of these Device Access Groups, then you can run the playbook on this device. The operation fails if the device does not belong to any of your Device Access Groups.
  - If you select multiple devices (using the **Allow bulk jobs** option or using tags) and if any of them does not have access, an error message appears stating that this list of devices does not have access to run the playbook.

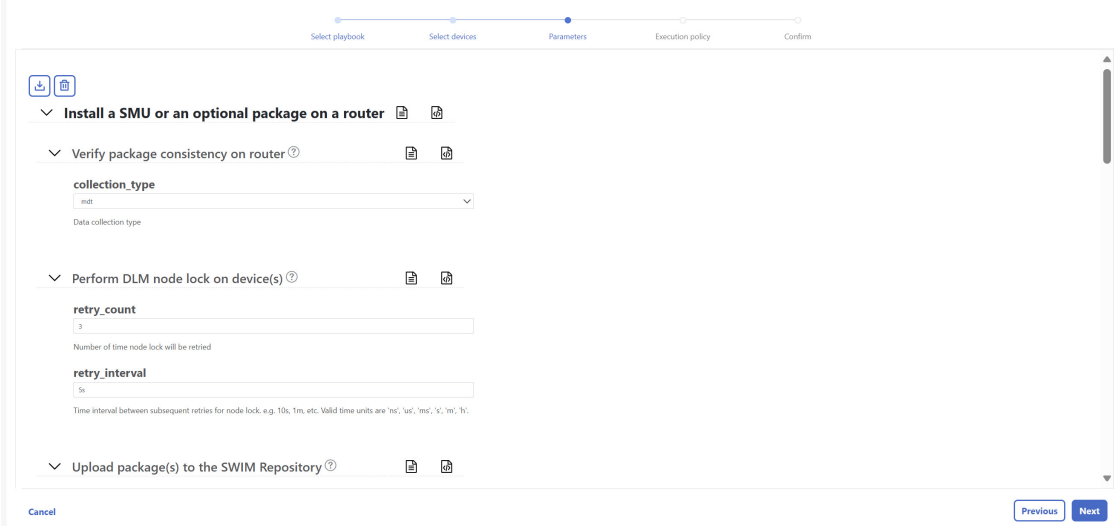
- In the **Select device manually** selection mode, you can check the **Allow bulk jobs** check box to select multiple devices and run the selected playbook on them simultaneously. Based on your selection, the system creates a static group of multiple jobs. Hover the mouse pointer over the ⓘ icon next to the check box for more information. There is no limit to the number of devices you can select for a bulk job.

  **Note**
  **Allow bulk jobs** option is enabled for playbooks that can be executed on a single device.

**Step 4** Click **Next**. The **Parameters** window appears.

**Step 5** In the fields provided in the **Parameters** window, enter the playbook parameter values for this run.

***Figure 28: Parameters***

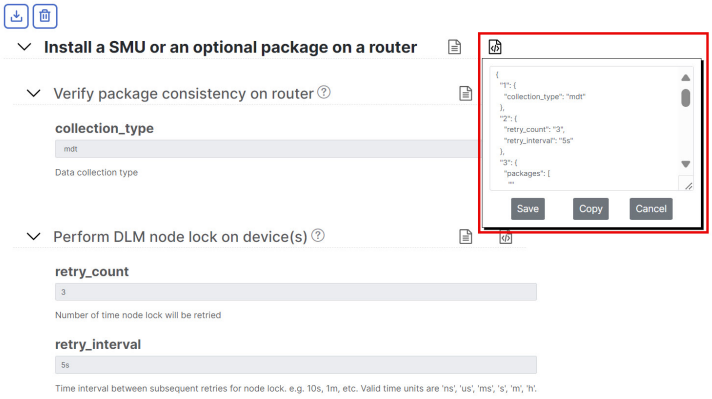



With the **Parameters** window displayed, you can also:

- Click ⬇ to upload a JSON file with the parameter values you want. You will be prompted to navigate to the JSON parameters file you have previously prepared (or downloaded from a previous playbook run) and then upload it as appropriate for your browser and operating system.

- Click ⟨/⟩ to enter the parameter values in JSON format. A popup text window displays the full list of JSON parameters with empty values in quotes. Edit the values, and when you are finished, click **Save**.

*Figure 29: Edit JSON*




- Click to select the items you want to configure. A pop-up text window will appear, displaying the complete list of plays or parameters that can be configured. For the playbook, it will show a list of plays and, for each selected play, it will display a list of parameters that can be configured for that play. If you unselect any item, it will not appear for configuration. Uneditable options indicate mandatory items that cannot be deselected.

*Figure 30: Object properties for a playbook*

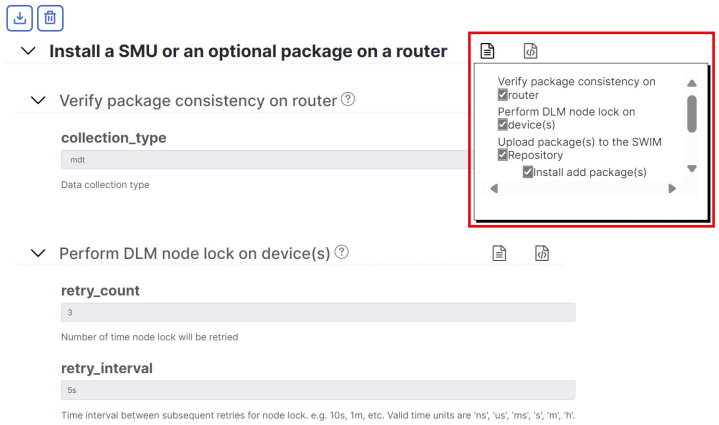
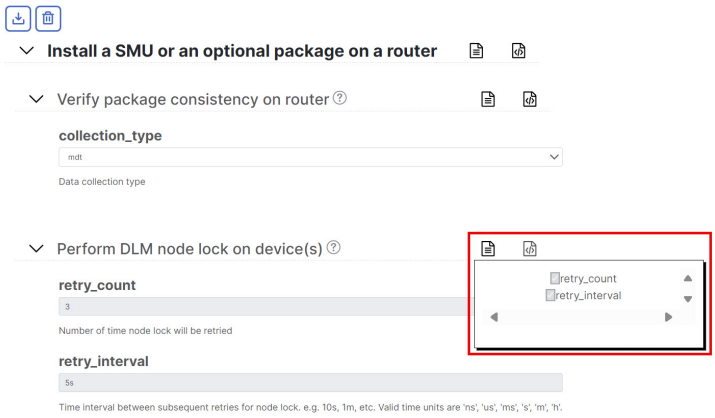


*Figure 31: Object properties for a play*

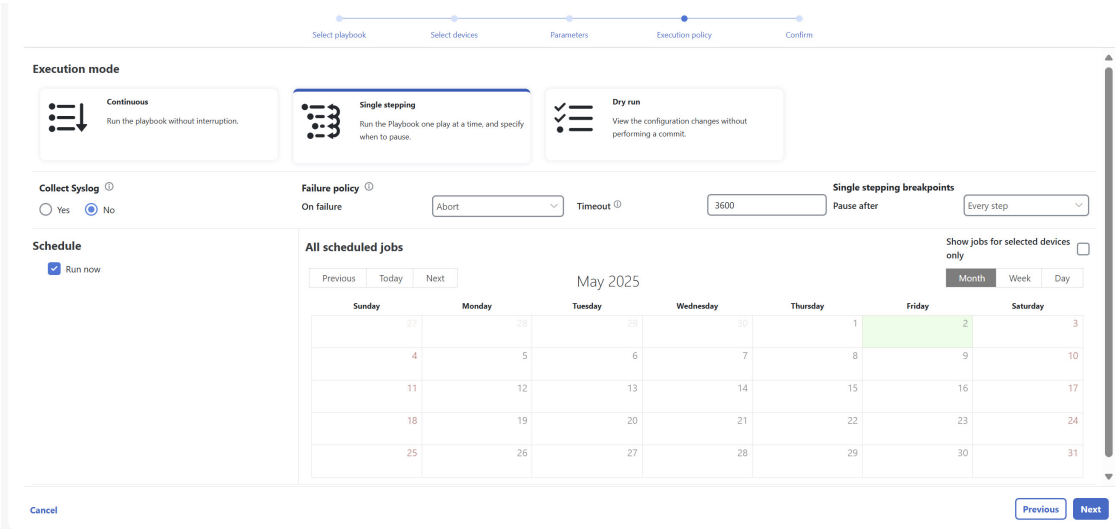



- Click ⊕ to add additional instances of a particular parameter, if required for the playbook you are running. Click 🗑 to delete instances added in this way.
- Click 🗑 to clear all the parameter values entered so far.

**Step 6** With the parameter values set, click **Next**. The **Execution policy** window appears.

**Step 7** Choose **Single stepping**. The **Execution policy** window displays additional features to customize the job:

*Figure 32: Execution policy*




- Under **Collect syslog**, click **Yes** if you want syslogs to be collected during and immediately after the run and **No** if you do not. **Yes** is the default selection only if you have a syslog provider configured.
- From the **Failure policy** dropdown, select:
  - **Abort** to abort the entire run without rolling back any changes, if a failure occurs at any point. This is the default. Any configuration changes made up to the point of failure will not be rolled back.

- **Pause** to pause the run and allow you to decide how to handle the failure. This pause will be in addition to any breakpoints you set using the **Single stepping breakpoints** dropdown.
- **Complete roll back** to abort the entire run and roll back all configuration changes made.

- In the **Schedule** area, uncheck the default **Run now** selection to schedule the job for a later time. See Schedule playbook runs, on page 40 for help on using the **Schedule** area features.

**Step 8** From the **Single stepping breakpoints** dropdown, select either

- **Every step** to pause automatically after every step in the playbook.
- **Customize** to select the steps where you want the playbook to pause.

If you select **Customize**, the **Customize check point** pop-up displays a list of all the plays in the playbook, with a ⏸ at the step between each play. Click the ⏸ at each step where you want to set a breakpoint. When you are finished, click **Done**.

**Step 9** Click **Next**. The **Review your job** window appears, displaying a summary of your choices: playbook, devices, parameters, and execution policy. In this window:

- You must provide a relevant **Name** for the job.
- You can enter labels for your job using the **Labels** field.
- You can click on any **Change** links in the **Review your job** window summary to modify your choices.

**Step 10** (Optional) Enter the device credentials (name and password).

**Note**
This step is applicable only if **Credential prompt** is enabled in the Change Automation settings. For more information, see Verify installation and configure system settings.

**Step 11** When you are ready to continue, click **Run playbook**.

**Step 12** At the confirmation prompt, click **Confirm**. Click **View job set** to view the status of the current job. The job details include job status, job set tags, the title of the selected playbook, execution parameters and policy, last updated date, and updated comments (if any).

**Step 13** While the run is executing, the **Running** tile at the top of the window will change to **Paused** for each step at which you have set a breakpoint. Your choices at each pause will be displayed as buttons below the tiles:

- Click **Resume** to resume running from this point, with no changes. The **Resume** request includes the runtime parameters from the previous step; you can edit these, as needed, later.
- Click **Roll back** to roll back any changes made so far. You can choose how far to rollback:
  - Click **Complete roll back** to roll back all changes to the start of the playbook run. Once you have rolled back to the start, you can choose to **Resume** from that point, **Abort** the run entirely, or **Edit runtime parameters** of the run.
  - Click **Select roll back point** to roll back changes to your selected step. All the previous steps will have a roll back point icon next to them. Click this icon for the step to which you want to roll back. Once you have selected the step, you can choose to **Resume** from that step, **Roll back** further, **Abort** the run entirely or **Edit runtime parameters**.
- Click **Abort** to abort the run entirely. No changes made will be rolled back.

- Click **Edit runtime parameters** to edit the parameters the run is using. You edit using a pop-up version of the **Parameters** window, just as you did in step 6. When resuming, the parameters exposed for editing are specific to the task being resumed, meaning they are not the same global parameters you defined in step 6. Most of the time, they are a subset of the global parameters. When you are finished, click **Apply**. You can then choose to **Resume** execution with the changed parameters.

**Step 14** While the run is executing, you can also use the following features of the progress window:

- View the execution status of each play in the playbook in the **Maintenance** play list at the left side of the window. plays that fail are indicated with a red icon; plays that succeed are indicated with a green icon.
- See reminders of your choices in the **Playbook** and **Devices** tiles at the top of the window.
- See the current status of the run in the **Running** tile at the top of the window.
- Click **View** in the **Parameters** tile to view the run's parameters. While viewing the parameters, you can click **Download parameters** to save them in a JSON file. You will be prompted to name and save the file appropriately for your browser and operating system.
- Use the network topology in the map at the right side of the window to view the device and its connections to the rest of your network.

**Step 15** After the run is complete:

- Click the **Events** tab to see success and failure messages for each step of the playbook. This can help you diagnose and correct problems with individual plays and the run as a whole.
- Click the **Syslog** tab to access syslog messages collected during and immediately after the run. If syslog collection is enabled, the tab will provide a pointer to the path on the syslog storage provider where collected syslogs are stored. If you chose not to collect syslogs, or no syslog storage provider has been configured, this tab will display a message indicating that syslog collection is disabled.
- Click the **Console** tab to see relevant commands and responses from the device consoles that took place during the run. These messages can also help with diagnostics.
- An event is created in the audit log (**Administration** > **Audit log**). The audit log includes details like the name of the playbook, the user who ran the playbook, and the commit label, if present.

# Run playbooks in continuous mode

Continuous execution mode is the standard way to run playbooks. Configuration changes are committed to the device during the run, with no checks or delays except those programmed for system resets or other purposes. The run continues until it succeeds or fails. If it fails, you can use the run's Failure Policy to abort, rollback all changes made to the device, or pause execution at the failure point.

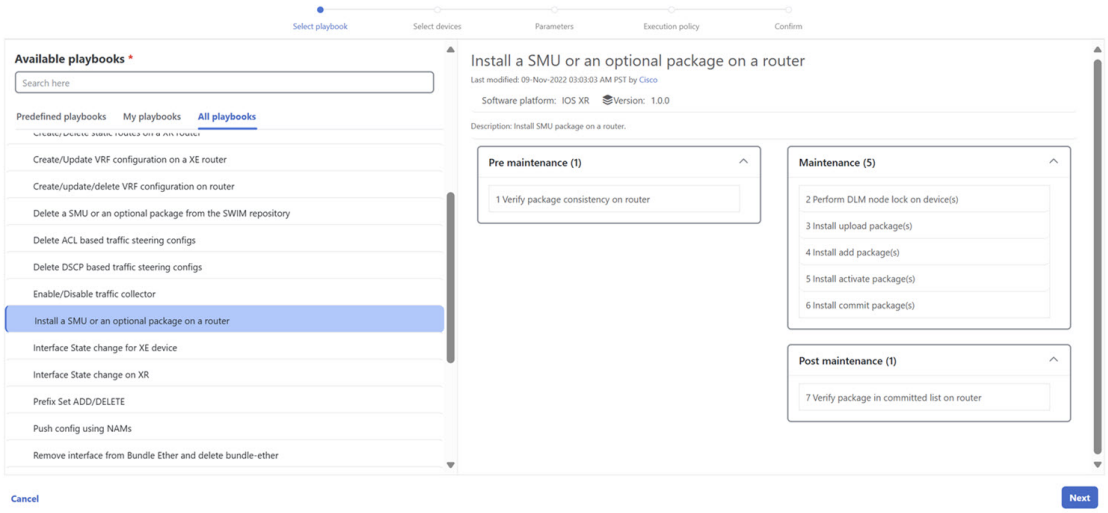
It is always good practice to perform a dry run and verify the configuration changes before committing to a continuous run (see Perform a dry run of a playbook, on page 22). You can also run the playbook in single-stepping mode, which will allow you to pause execution after any play you select, abort and rollback changes as needed, and even change runtime parameters in the middle of the run (see Run playbooks in single stepping mode, on page 28).

## Procedure

**Step 1** From the main menu, choose **Network Automation** > **Run playbook**.

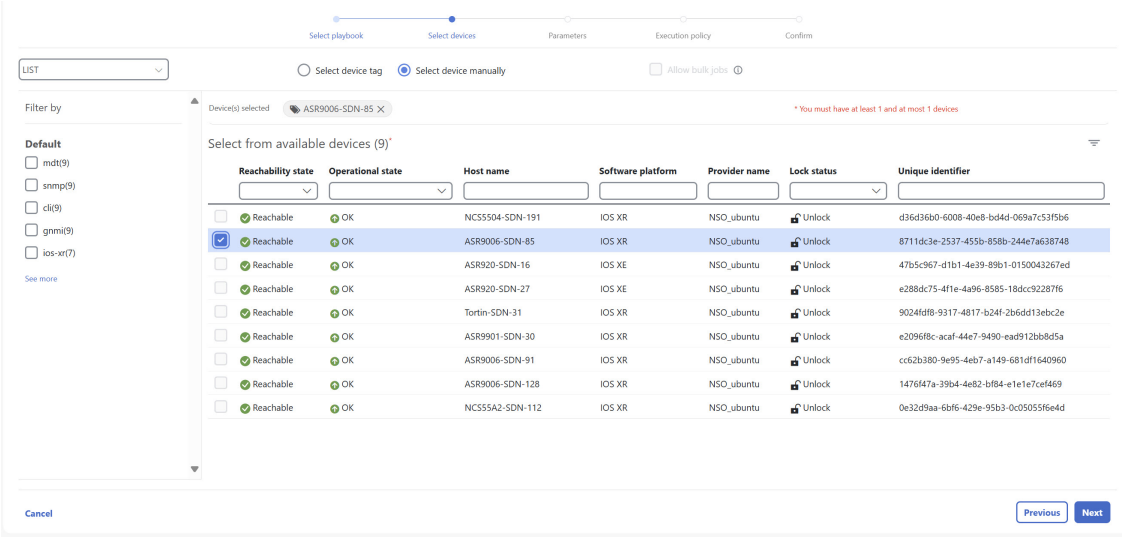
**Step 2** In the **Available playbooks** list on the left, click on the playbook you want to run. On the right, the window displays the playbook name, hardware and software compatibility information, and descriptions for all the plays in the selected playbook.

***Figure 33: Select playbook***




**Step 3** Click **Next**. The **Select devices** window appears.

***Figure 34: Select devices***




Using this window:

- You can toggle between the table view and topology map view by clicking and selecting the relevant option in the drop-down button on the upper left corner of the window. Choose **Select devices from list** or **Select devices from map** to select the table view or topology map view, respectively. By default, the table view is displayed.
- With the topology map view displayed, you can toggle between the map's geographical and logical views by clicking on the or the . You can also zoom, display bandwidth utilization, and change logical view layouts as you do with the topology map you see when you select **Topology** from the main menu.
- You can select the devices manually or using tags. The **Select device tag** option targets you to select the relevant tag instead of devices from the table and all devices associated with the relevant tag are selected. The **Select device manually** option allows you to select the devices from the list using quick and advanced filters and filter by tags on the left. Hover the mouse pointer over the icon next to the options for more information. You can also view the selection criteria such as number of devices required for the selected playbook.

  **Note**
  If you are a non-admin user and selecting the devices manually, make a note of the following:

  - The devices on which you want to run the playbook must belong to a Device Access Group and you must have access to this Device Access Group. For more information on Device Access Groups and associating a user with a Device Access Group, see the Manage Device Access Groups section in the *Cisco Crosswork Network Controller Administration* guide.
  - If your role is associated with an empty Device Access Group, you will receive an error message.
  - If your role is associated with multiple Device Access Groups and if the device belongs to any one of these Device Access Groups, then you can run the playbook on this device. If the device does not belong to any of your Device Access Groups, the operation fails.
  - If you are selecting multiple devices (using **Allow bulk jobs** option or using tags) and if any devices does not have access, an error message appears stating that this list of devices does not have access to run the playbook.

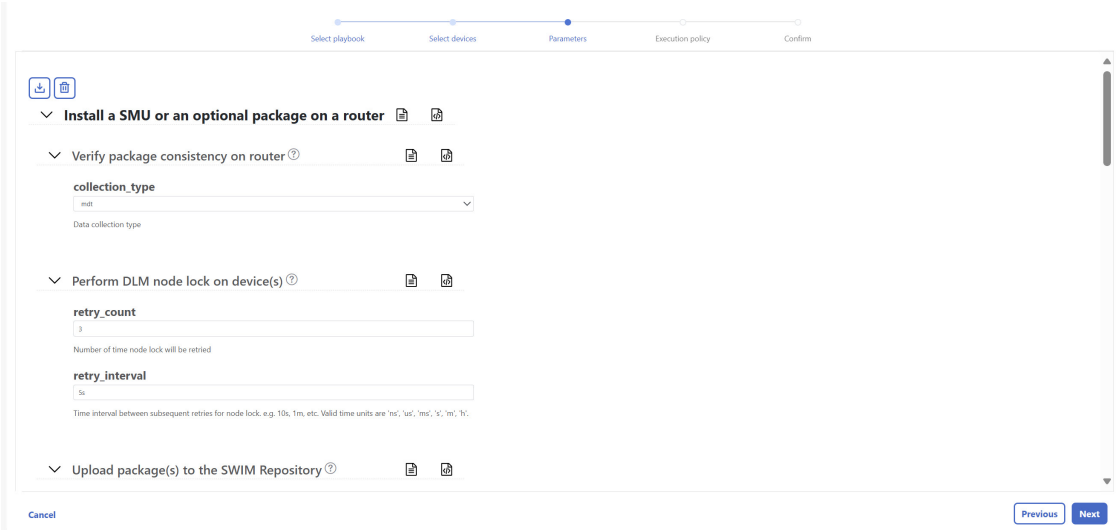
- In the **Select device manually** selection mode, you can check the **Allow bulk jobs** check box to select multiple devices and run the selected playbook simultaneously. Based on your selection, the system creates a static group of multiple jobs. Hover the mouse pointer over the icon next to the check box for more information. There is no limit to the number of devices you can select for a bulk job.

  **Note**
  **Allow Bulk Jobs** option is enabled for playbooks that can be executed on a single device.

**Step 4** Click **Next**. The **Parameters** window appears.

**Step 5** In the fields provided in the **Parameters** window, enter the playbook parameter values to use for this run.

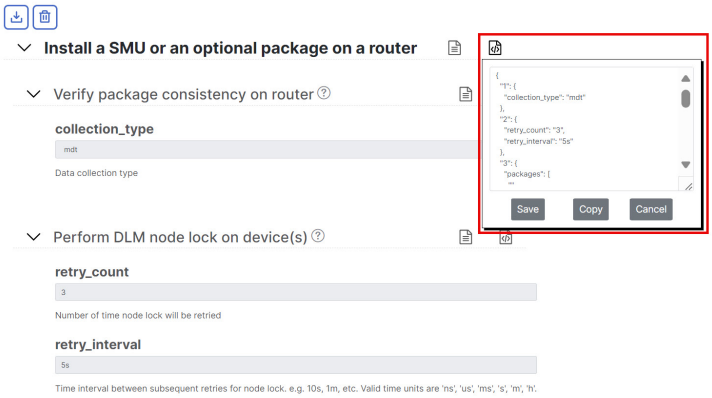
*Figure 35: Parameters*




With the **Parameters** window displayed, you can also:

- Click to upload a JSON file with the parameter values you want. You will be prompted to navigate to the JSON parameters file you have previously prepared (or downloaded from a previous playbook run) and then upload it as appropriate for your browser and operating system.
- Click to enter the parameter values in JSON format. A popup text window displays the full list of JSON parameters with empty values in quotes. Edit the values, and when you are finished, click **Save**.

  *Figure 36: Edit JSON*




- Click to select the items you want to configure. A pop-up text window will appear, displaying the complete list of plays or parameters that can be configured. For the playbook, it will show a list of plays and, for each selected play, it will display a list of parameters that can be configured for that play. If you unselect any item, it will not appear for configuration. Uneditable options indicate mandatory items that cannot be deselected.

*Figure 37: Object properties for a playbook*

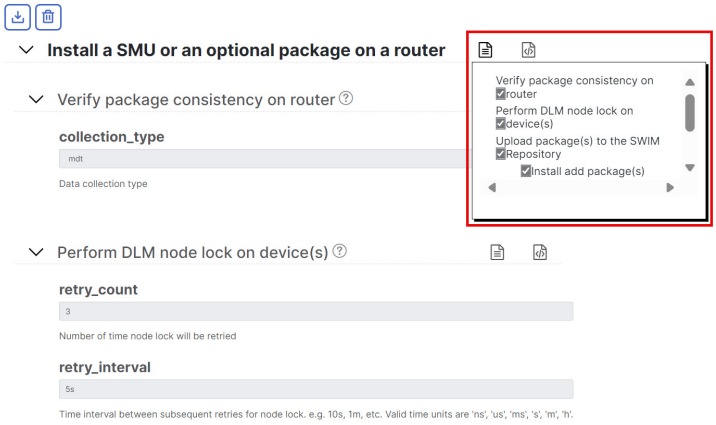
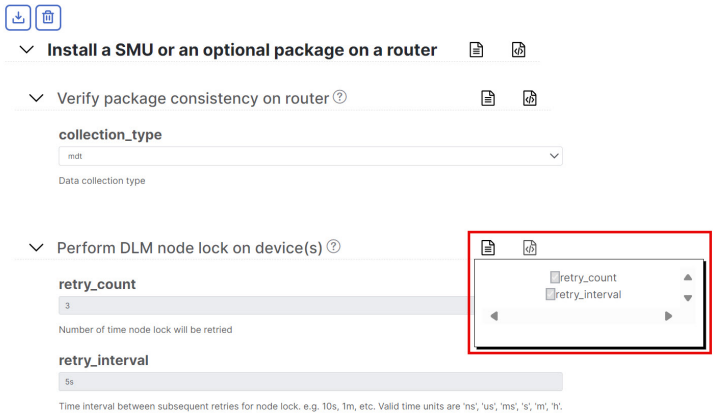



*Figure 38: Object properties for a play*

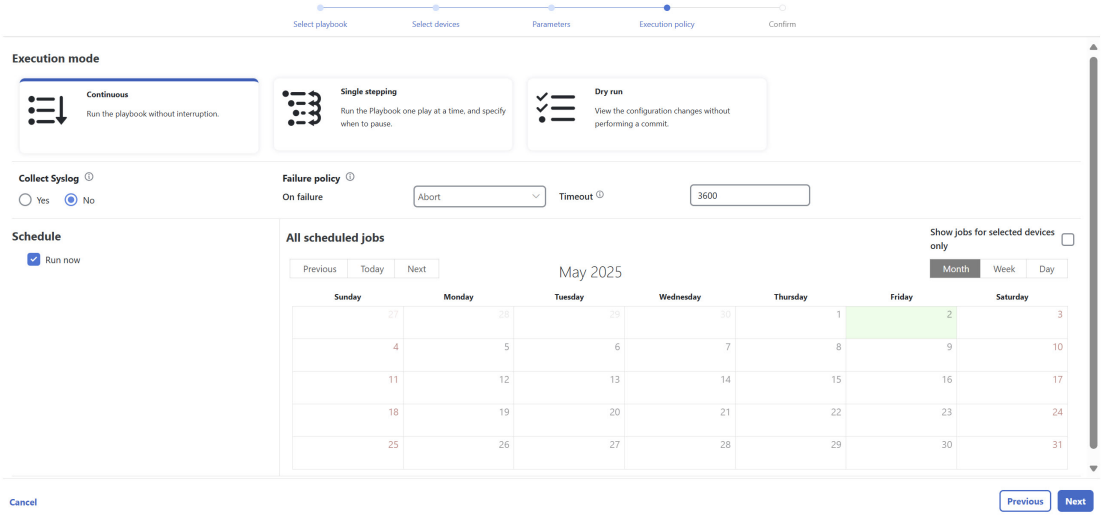



- Click ⊕ to add additional instances of a particular parameter, if required for the playbook you are running. Click 🗑 to delete instances added in this way.
- Click 🗑 to clear all the parameter values entered so far.

**Step 6** With the parameter values set, click **Next**. The **Execution policy** window appears.

**Step 7** Choose **Continuous**. The **Execution policy** window displays additional features to customize the job:

*Figure 39: Execution policy*




- Under **Collect syslog**, click **Yes** if you want syslogs to be collected during and immediately after the run, and **No** if you do not. **Yes** is the default selection only if you have a syslog provider configured.
- From the **Failure policy** dropdown, select:
  - **Abort** to abort the entire run without rolling back any changes, if a failure occurs at any point. This is the default. Any configuration changes made up to the point of failure will not be rolled back.
  - **Pause** to pause the run and allow you to decide how to handle the failure.
  - **Complete roll back** to abort the entire run and roll back all configuration changes made.
- In the **Schedule** area, uncheck the default **Run now** selection to schedule the job for a later time. See Schedule playbook runs, on page 40 for help on using the **Schedule** area features.

**Step 8** Click **Next**. The **Review your job** window appears, displaying a summary of your choices: playbook, devices, parameters, and execution policy. In this window:

- You must provide a relevant **Name** for the job.
- You can enter labels for your job using the **Labels** field.
- You can click on the **Change** links in the **Review your job** window summary to modify your choices.

**Step 9** (Optional) Enter the device credentials (name and password).

**Note**
This step is applicable only if **Credential prompt** is enabled in the Change Automation settings. For more information, see Verify installation and configure system settings.

**Step 10** When you are ready to continue, click **Run playbook**.

**Step 11** At the confirmation prompt, click **Confirm**. Click **View job set** to view the status of the current job. The job details include information such as job status, job set tags, the title of the selected playbook, execution parameters, and policy, last updated date, and update comments (if any).

**Step 12** While the run is executing, the **Running** tile at the top of the window will change to **Paused** if you chose a **Failure policy** of **Pause**. Your choices will be displayed as buttons below the tiles:

- Click **Resume** to resume running from this point, with no changes.
- Click **Roll back** to roll back any changes made so far.
- Click **Abort** to abort the run entirely. No changes made will be rolled back.

**Step 13** While the run is executing, you can also use the following features of the progress window:

- View the execution status of each play in the playbook in the **Maintenance** play list at the left side of the window. Plays that fail are indicated with a red icon; plays that succeed are indicated with a green icon.
- See reminders of your choices in the **Playbook** and **Devices** tiles at the top of the window.
- See the current status of the run in the **Running** tile at the top of the window.
- Click **View** in the **Parameters** tile to view the run's parameters. While viewing the parameters, you can click **Download parameters** to save them in a JSON file. You will be prompted to name and save the file as appropriate for your browser and operating system.
- Use the network topology in the map at the right side of the window to view the device and its connections to the rest of your network.

**Step 14** After the run is complete:

- Click the **Events** tab to see success and failure messages for each step of the playbook. This can help you diagnose and correct problems with individual plays and the run as a whole.
- Click the **Syslog** tab to access syslog messages collected during and immediately after the run. If syslog collection is enabled, the tab will provide a pointer to the path on the syslog storage provider where collected syslogs are stored. If you chose not to collect syslogs, or no syslog storage provider has been configured, this tab will display a message indicating that syslog collection is disabled.
- Click the **Console** tab to see relevant commands and responses from the device consoles that took place during the run. These messages can also help with diagnostics.
- An event is created in the audit log (**Administration** > **Audit log**). The audit log includes details like the name of the playbook, the user who ran the playbook, and the commit label, if present.

# Schedule playbook runs

The Change Automation application's **Execution mode** window allows you to schedule future playbook runs as jobs and view all the jobs that have been scheduled. Use the **Schedule** area on the left to schedule a job. Use the **All scheduled jobs** area on the right to view scheduled jobs on the calendar.

**Note** **Playbook job scheduling** is only available if enabled when Change Automation was installed and initially configured. For more information, see Verify installation and configure system settings. To change this setting, you must uninstall and then reinstall Change Automation.

**Note** If you are a non-admin user, ensure you have access to the Schedule playbook task. You cannot schedule playbooks without this task.

**Prerequisites**:

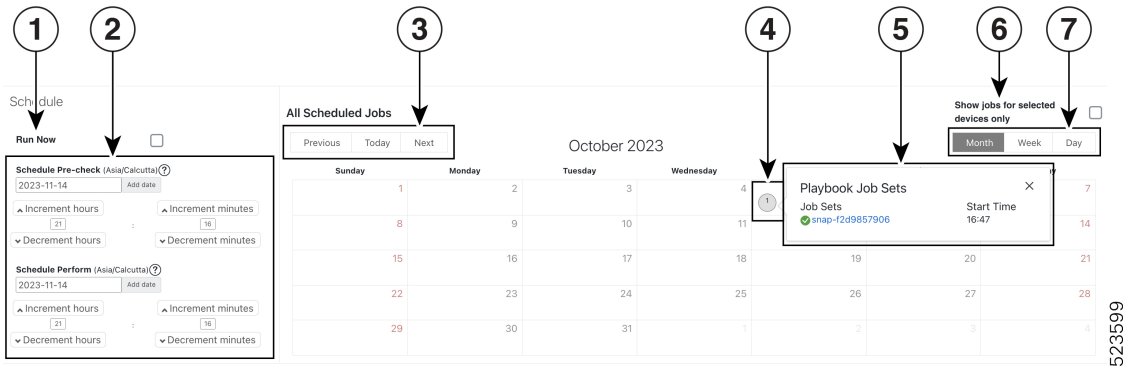
Ensure that **Playbook job scheduling** is enabled on the Device Override Credentials page. For more information, see Verify installation and configure system settings.

To enable the task permission, do the following:

1. Go to **Administration** > **Users and Roles** > **Roles**.
2. Under the **Roles** pane, select the role for which you want to grant the access.
3. Under the **Task permissions** tab, enable the **Schedule playbook** check box and click **Save**.

The **Execution mode** window's scheduling features are only displayed when you have chosen to run a playbook in continuous or single-stepping mode. You cannot schedule a dry run of a playbook.

***Figure 40: Execution mode scheduling features***




| Item | Description |
|---|---|
| 1 | **Run now**: Running playbooks immediately is the default for continuous and single-stepping execution modes. To schedule a run for a future time and date, you must uncheck this box. |
| 2 | **Schedule selectors**: Use these fields to select the future time and date when the playbook runs. Although it is the default for the Pre-Maintenance and Maintenance phases of a scheduled playbook to start simultaneously, you can use the upper **Schedule pre-check** and lower **Schedule perform** fields to schedule the start of Pre-Maintenance and the start of Maintenance independently. The **Schedule perform** time must always be greater than or equal to the **Schedule pre-check** time. |
| 3 | **Previous/Today/Next selectors**: Use these three selectors with the **Month/Week/Day** selectors to focus the calendar's display of scheduled jobs on the time range in which you are interested. For example: To show only those jobs scheduled for next week, click **Next** and **Week**. |
| 4 | **Job icons**: Red, numbered icons in the squares representing each calendar date show how many jobs are scheduled for that date. Yellow circle icons represent each scheduled job. |

| Item | Description |
|---|---|
| 5 | **Job details pop-up**: Hover your mouse cursor over a yellow circle icon to see the details for the scheduled job represented by that icon. The pop-up shows the execution ID of the job and the name of the playbook to be run. |
| 6 | **Show jobs for selected devices only**: Check this box to restrict the calendar display to only jobs scheduled to run on the devices you have already selected. This is a handy way to see if the schedule you plan for your playbook run conflicts with other scheduled jobs on the same devices. |
| 7 | **Month/Week/Day selectors**: Use these three selectors with the **Previous/Today/Next** selectors to focus the calendar's display of scheduled jobs on the time range in which you are interested. For example: To show only those jobs scheduled for last month, click **Last** and **Month**. |

**Note** Change Automation playbooks have a **`mop_timeout`** parameter, which is a user-specified input needed to schedule any playbook.

If you are scheduling a playbook with **Failure policy** set to **`Complete Roll Back`**, first dry run the play and note the time taken. Then, add a buffer time (for example, 10 minutes) to the time taken during the dry run. After that, double the time value and enter it to the **`mop_timeout`** parameter, as it can take as much time to roll back the playbook as it takes to run it until the last step. Without sufficient **`mop_timeout`**, the playbook can end up incomplete (in between transitions) if the timeout gets triggered while rollback is in progress. If this happens, you have to revert the changes manually or create a playbook with the changes you want to revert.

# View or abort playbook jobs

The **Automation Job History** window lets you click on any individual job in the list to see that job's detailed execution progress panel. This panel displays the name of the playbook, its plays, the devices it ran on, the parameters used, and all events, Syslog, console, and other messages. These details are helpful when diagnosing failures.

The **Automation Job History** window also allows you to abort *running* jobs.

You can also navigate to **Automation Job History** window from the **Jobs** panel in the Change Automation **Dashboard**.

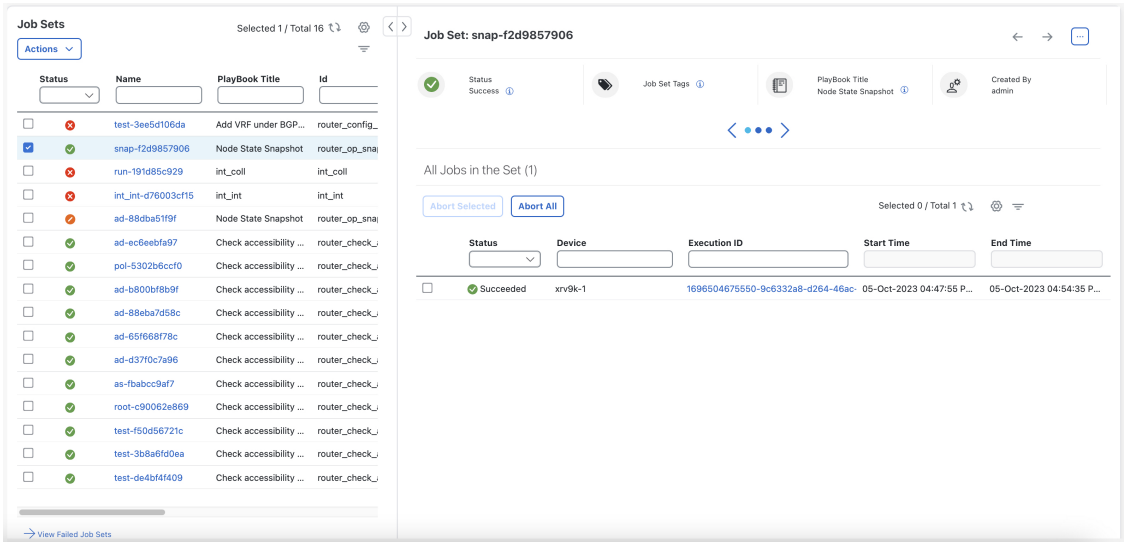
### Before you begin

A user must have the permission for specific playbook label to run or abort a playbook. For more information on assigning playbooks to specific roles, see Assign playbooks to specific roles, on page 20.

### Procedure

**Step 1** From the main menu, select **Network Automation** > **Automation Job History**. The **Automation Job History** window displays a list of Job Sets.

*Figure 41: Automation Job History*




The list in **Automation Job History** window is sorted by the last update time, with running or most recently executed jobs at the top. You can apply quick or advanced filters to the table as you would with columns in other table windows.

**Step 2** To view information about a playbook job, click the relevant job ID checkbox on the left. The job's status and execution details are displayed on the right side. Click on the ⓘ icon next to each detail to get more information about the selected job set.

**Step 3** You can abort a job set in running, paused or scheduled status, as follows:

- To abort a specific job, click the check box next to it and then click **Abort selected**.
- To abort all jobs immediately, click **Abort all**.

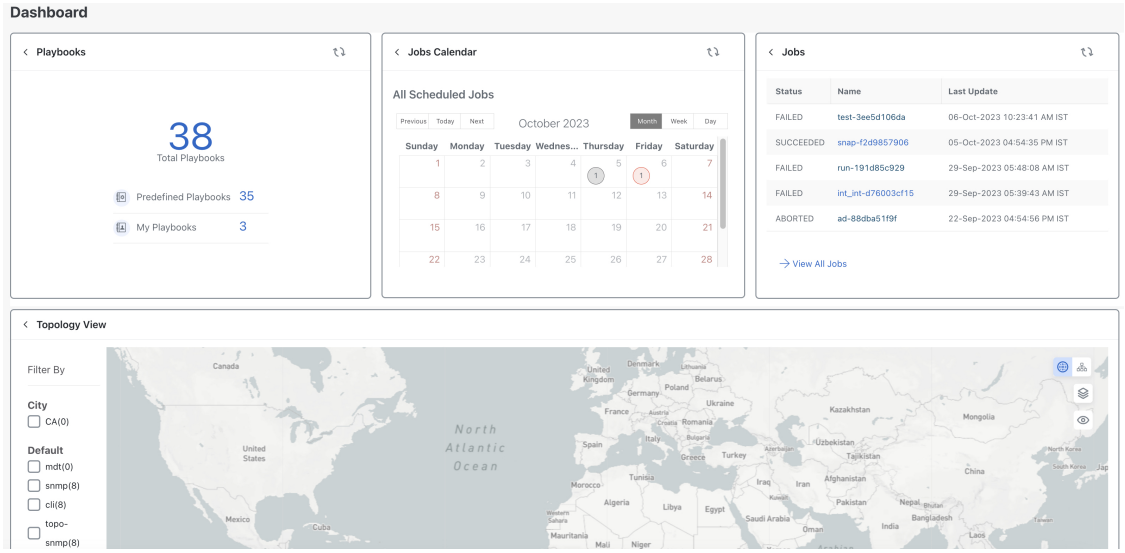
When prompted, click **Confirm**. Jobs currently running, paused, or scheduled will abort once the current task has been completed.

# Use the change automation dashboard

The Change Automation **Dashboard** window (shown in the figure below) lets you view all playbook-related activity and initiate playbook runs. It displays the total number of playbooks, the playbook Jobs Calendar, the most recently run playbook jobs, and the same network topology map you see when you select **Topology** from the main menu.

To view the Change Automation **Dashboard** window, select **Network Automation** > **Dashboard**.

*Figure 42: Change Automation Dashboard Window*




The **Playbooks** tile displays the total number of playbooks (pre-defined and custom). Clicking on a specific number displays all the playbooks that correspond to the selected category:

- **Total playbooks** indicate the total number of pre-defined and user-created playbooks (My playbooks) in the system.
- **Predefined playbooks** indicate the number of pre-defined playbooks that exist in the system.
- **My playbooks** indicate the number of custom playbooks created by the current user.

Creating playbooks does not use a license. The license count is incremented only upon the first execution of a playbook (pre-defined or user-created), irrespective of whether the playbook runs successfully. Subsequent execution of the playbook does not increment the license count.

The **Jobs calendar** tile displays a calendar (month, week, day) with the number of job sets executed on a given day marked in a circle against the corresponding date. Clicking on the number displays a dialog box with the names of the playbook job sets and their execution time. Click the desired job set to view the execution details.

The color of the circle indicates the overall status of the job sets:

- A **red** circle indicates at least one job set with **Failed** status among the day's overall job sets.
- A **gray** circle indicates that all job sets are in **Scheduled** or **Running** status.
- A **blue** circle indicates at least one critical job set in **Recovered** status among the day's overall job sets.
- A **green** circle indicates most of the playbooks are in success state. Clicking on it displays all the jobs that are **Recovered**, **Scheduled**, or **Running**.

The **View All Jobs** link on the **Jobs** tile gives you direct access to the Change Automation **Automation Job History**.

# Troubleshoot change automation

The following table describes issues you may encounter when using the Change Automation application and their solutions or workarounds.

***Table 1: Change Automation Troubleshooting***

| Issue | Solution |
|---|---|
| playbook run fails with messages indicating that Cisco Network Services Orchestrator (Cisco NSO) and the target device are out of sync or communication. Message text varies but may include "device out of sync," "NC client timeout," and other text indicating connectivity or sync issues between Cisco NSO and the device. | Ensure that the playbook does not include a sync operation. Get the device and Cisco NSO back in sync, and then re-run the playbook.<br>Alternatively, you can create a new playbook that includes a sync operation to avoid future problems. |
| playbook run fails with "access error" messages indicating failure "to set device override credentials in NSO". | Ensure that "admin" is a member of the **ncsadmin** user group in Cisco NSO. |
| *"Failed to end NSO transaction, 500:fatal:YClientError: Failed to send RPC:"* error is displayed while running the playbook. | Include the below settings in the Cisco NSO configuration file (`ncs.conf`):<br>`<ssh>`<br>`<client-alive-interval>infinity</client-alive-interval>`<br>`<client-alive-count-max>5</client-alive-count-max>`<br>`</ssh>`<br>**Note**<br>This configuration could increase the load on Cisco NSO, so it is better to do it only when necessary. |
| Playbook aborted due to failure in locking the device nodes. | In the Devices window, select the relevant devices and clear the lock by moving the device to DOWN and then UP. Go to **Administration** > **Crosswork Manager**, click the **Change Automation** tile, and restart the robot-nca process. Once the protocols are reachable, you can schedule to run a new playbook. |
| SMU install fails at "Verify package in committed list on router". | Instead of using the tar.gz file in the packages field under Verify package in the committed list on router sub-option, use the committed package name to verify the package. |