HP Cloud Connector

For the Windows ® Linux operating systems

Software Version: 1.10

HP Cloud Connector Help

Legal Notices

Warranty

The only warranties for HP products and services are set forth in the express warranty statements accompanying such products and services. Nothing herein should be construed as constituting an additional warranty. HP shall not be liable for technical or editorial errors or omissions contained herein.

The information contained herein is subject to change without notice.

Restricted Rights Legend

Copyright Notice

Trademark Notices

Adobe® is a trademark of Adobe Systems Incorporated.

Microsoft® and Windows® are U.S. registered trademarks of Microsoft Corporation.

UNIX® is a registered trademark of The Open Group.

Documentation Updates

The title page of this document contains the following identifying information:

- Software Version number, which indicates the software version.

- Document Release Date, which changes each time the document is updated.

- Software Release Date, which indicates the release date of this version of the software.

To check for recent updates or to verify that you are using the most recent edition of a document, go to:

http://h20230.www2.hp.com/selfsolve/manuals

This site requires that you register for an HP Passport and sign in. To register for an HP Passport ID, go to:

http://h20229.www2.hp.com/passport-registration.html

Or click the **New users - please register** link on the HP Passport login page.

You will also receive updated or new editions if you subscribe to the appropriate product support service. Contact your HP sales representative for details.

Support

Visit the HP Software Support Online web site at:

http://www.hp.com/go/hpsoftwaresupport

This web site provides contact information and details about the products, services, and support that HP Software offers.

HP Software online support provides customer self-solve capabilities. It provides a fast and efficient way to access interactive technical support tools needed to manage your business. As a valued support customer, you can benefit by using the support web site to:

- Search for knowledge documents of interest

- Submit and track support cases and enhancement requests

- Download software patches

- Manage support contracts

- Look up HP support contacts

- Review information about available services

- Enter into discussions with other software customers

- Research and register for software training

Most of the support areas require that you register as an HP Passport user and sign in. Many also require a support contract. To register for an HP Passport ID, go to:

http://h20229.www2.hp.com/passport-registration.html

To find more information about access levels, go to:

http://h20230.www2.hp.com/new_access_levels.jsp

Disclaimer for PDF Version of Online Help

This document is a PDF version of the online help. This PDF file is provided so you can easily print multiple topics from the help information or read the online help in PDF format.

Note: Some topics do not convert properly to PDF, causing format problems. Some elements of online help are completely removed from the PDF version. Those problem topics can be successfully printed from within the online help.

Contents

Welcome to HP Cloud Connector Help

The Cloud Administration Dashboard is used to view, allocate, and manage all virtual resources within a domain. This dashboard provides a simplified means of creating and managing Cloud Connector resources, infrastructure topology templates, resource binding documents, as well as the ability to provision a composite topology described by the topology templates and binding documents. Cloud Connector launches resource binding documents to create virtual infrastructure in the cloud. Cloud Administration Dashboard is associated with a single domain.

The Cloud Installation Dashboard is used to install, set up, and start a cloud environment which is then managed through the Cloud Administration Dashboard.

Welcome to HP CDA Topology Designer Help

HP Continuous Delivery Automation documentation for the Topology Designer is available on the HP Software Product Manuals website at *http://h20230.www2.hp.com/selfsolve/manuals*.

- General-access documentation requires that you register for an HP Passport and sign-in.

- Restricted-access documentation requires that you have an active HP support agreement ID (SAID) and an HP Passport sign-in.

Welcome to HP Cloud Administration Dashboard Help

Cloud Installation Dashboard is the installer that is used to install, set up, and start a cloud environment with the Cloud Administration Dashboard.

The Cloud Administration Dashboard is used to view, allocate, and manage all virtual resources within a domain. Cloud Administration Dashboard provides a simplified means of creating and managing Cloud Connector resources, infrastructure topology templates, resource binding documents, as well as the ability to provision a composite topology described by the topology templates and binding documents. Cloud Connector launches resource binding documents to create virtual infrastructure in the cloud. Cloud Administration Dashboard is associated with a single domain.

Domains

A **domain** is a security grouping that provides context for the definition of Projects, Users, and Roles and hosts one or more regions. All users belong to a domain. Users are then linked to a region and projects through roles assigned by an Administrator.

Keystone is the security context and the defining service for the domain. All services connected to a single Keystone service are in one domain. Glance, in the recommend configuration, is also a domain service as the images it provides are available to VMs in all regions. Cloud Connector domain services are Eve, Focus, Peer, and Eden. A domain service is a service that can be used by or for any region, so all of the Cloud Connector services are considered domain services because they can be used for one or more regions.

Regions

Domains are divided into Regions. A **Region** is a compute grouping that provides context for the usage of compute servers, networking systems, and storage resources. A Region contains a defined set of users, projects, topology templates, infrastructure designs, and services governed by a single Cloud Connector Keystone domain service. Regions are used to group different locations, different types of hardware, or to separate different resources belonging to different groups.

Region services are registered under the names:

- Nova - A region is constructed by installing and naming the Cloud Connector cloud compute service called Nova.

- Volume - The creation and management of volumes is controlled by the Volume service which is accessed through the Nova service interface.

- EC2 - An alternate interface for the Nova service.

Region services can be run on the same node as the domain controller or they can be run on a different server. There can be multiple regions within a domain.

Projects

 Projects are groupings that allow users to view the same set of resources. Projects are the unit of ownership within Cloud Connector that protect topology templates, infrastructure designs, and regional resources from being visible to all users.

Access to the Cloud Administration Dashboard

The dashboard provides security by roles. See "Role Descriptions" on page 20.

Domain Tab - Administrator Only	
Services	Domain Services provides a read-only list of services running in the domain.
Projects	Projects are created by an administrator and allow a group of users to view the same set of resources. Within this area you can manage projects, add users as an administrator or user to a project, and manage user roles.
Users	Allows you to create new users and update, disable, and delete existing users.
Roles	Allows you to assign specific roles to a user within a region and project.

Region Tab - Administrator (All) and User (only sees Flavors and Limits)	
Overview	A view of region usage by month (defaults to the current month).
Services	A list of services and endpoints that are running in the region.
Instances	A list of all VMs running within the region.
Volumes	All of the disc storage volumes that have been allocated to the region.
Networks	Fixed IP Networks and Floating IP Pools.
Flavors	A list of machine configurations that can be used to create new VMs. A flavor describes the amount of memory, number of CPUs, and ephemeral disk space available to the VM.
Limits	A view of Default Absolute Limits and Default Rate Limits.

Project Tab - All	
Overview	Displays a Project Usage Summary.
Instances	A list of all VMs running within the project.

Project Tab - All	
Volumes	All of the disc storage volumes that have been allocated to the project.
Images	Images are all the launchable images (AMI, QCOW2, and RAW format) that can be used to create new VMs.
Snapshots	Snapshots preserve the disk state of running Instances and Volumes.
Access & Security	This tab provides a way to view and manage the following elements in the project: • Floating IPs • Security Groups • Keypairs
Topologies	Topologies represent the virtual infrastructures that have been realized when an Infrastructure Design document was launched in the Cloud Installation Dashboard.
Documents	Documents are the infrastructure topology templates and resource binding documents (design documents) that are used for provisioning.
Resource Pools	Records that define how to access a particular cloud provider.

Settings

The Settings link is available to all users on any window in the Cloud Administration Dashboard.

To change user display settings

1. Click the **Settings** link in the upper corner of a window.

2. In the left navigation frame, click **User Settings**.

3. **Timezone** defaults to UTC (Coordinated Universal Time). Click (⏷) for additional selections.

4. **Table page size** defaults to 20, meaning a maximum of 20 line items display at one time. Enter a different number if desired.

5. Click **Save**.

To view Cloud Connector API/CLI information

1. In the left navigation frame, click **Cloud Connector API/CLI**.

2. **API Keys** are properties that correspond to the **Current Place** login information (Domain/Region/Project) and user at the top of the window.

Item	Description
Tenant ID	Project ID.
Domain URL	Public URL for the domain security service.
Region ID	The name of the region you are logged into.
x-auth-token	Security token used to access service APIs. This token has a limited lifetime and is different every time you log into Cloud Connector.

3. **API Endpoints** lists the installed services you have access to and where their API endpoints are located for the logged in user.

4. Cloud Connector allows you to download the configuration file with environment variables for the current Domain, Region, and Project. When you click the **Download RC File** button the system creates a text file.

To Download EC2 Credentials

Credentials used with the EC2 interface to Cloud Connector.

1. In the left navigation frame, **click EC2 Credentials**.

2. Click the **Download EC2 Credentials** button to create a zip file that includes an RC file with your access/secret keys and your x509 private key and certificate.

To view Cloud Connector version

In the left navigation frame, click **About** to identify version.

Role Descriptions

All users belong to a domain. Users are then linked to a region and projects through roles assigned by an Administrator.

Within Cloud Connector, there are four basic roles:

- Domain Administrator

- Domain Architect

- Project Administrator

- Project User

Domain Level Roles

- **Domain Administrator**- all administrator rights associated with the cloud and all Cloud Connector services, including the ability to make resources available to users.

- **Domain Architect**- ability to create images, flavors, and topology components (resource binding documents, templates, etc.) so that the domain administrator can make them available to users.

Project Level Roles

- **Project Administrator** - owns all resources within a project and has as full access to modify and delete resources. Multiple users can be assigned the role of Project Administrator.

- **Project User** - owns only the content they have created. A Project User can modify or delete the content they own. All project users can view all content within their project even if owned by other users. Users outside of a project can see content only if it is marked as public.

You can also add additional Project Roles to a user. See "Add a Project Role to a User" on page 25 for descriptions of the following roles:

- binding_designer

- compute_admin

- compute_user

- document_publisher

- image_consumer

- image_creator

- image_publisher

- job_consumer

- job_creator

- rp_consumer

- rp_creator

- service_map_consumer

- template_designer

- topology_creator

General rules about content ownership within projects and regions

- To create content within a project, a user must be associated with the project and have either an administrative role or a specific content-creation role. The Project User role includes content-creation roles for most types of resources.

- Content can only be made Public by a user with an administrative role or a specific content-publishing role.

- Content belonging to a project is visible to all users associated to that project only if it has been made Public by an administrator. Only the latest version of a document is made public.

- While a project administrator can modify any content belonging to a project, a non-administrator can modify only the content created by that user.

Domains

The **Domain** tab within Cloud Connector is only available to a Domain Administrator. A domain is a security grouping that provides context for the definition of Projects, Users, and Roles. A domain is constructed by installing and naming a Keystone service. Keystone provides user authentication and tracks projects for all regions in a domain.

See the following related topics:

- "View Domain Services" below

- "View Domain Projects" on next page

- "View Domain Users" on page 31

- "View Domain Roles" on page 34

View Domain Services

Domain Services displays a read-only list of services that are registered in the domain and allows you to view the Service Map for selected services.

To view domain services

1. In the left navigation frame of the **Domain** tab, click **Services**. The following domain services are listed:

Service Name	Service Type	Description
Keystone	Identity and security	Provides user authentication and tracks various projects within the regions in the domain. If Keystone is disabled, you cannot log into the Cloud Administration Dashboard.
Glance	Image storage	Repository for images used to create new VMs.
Eve	Provisioning	A composite provisioning service that uses a topology (topology template and resource binding) to create VMs, volumes, and network segments as described in the topology. Eve uses the Peer service for determining resource pool existence, traits, and user identity mapping.
Focus	Template storage	Repository for infrastructure topology templates and resource binding documents that are used for provisioning.

Service Name	Service Type	Description
Peer	Resource Pool registry	Provides a registry of resource pool definitions you can use to provision to various cloud providers.
Eden	Framework and Utility Service	This service provides core functionality such as authorization and job access. The framework allows scaling of services so they can be distributed across multiple machines if needed. All Cloud Connector services are built and supported on the Eden framework.

2. The **Service URL (Public)** column shows the IP address where the service is running.

3. **Status** - Enabled

To view a service map

If the Service Name is a blue link, click to open the Service Map.

The Service Map helps an administrator determine the health of the system by showing each component that supports a service, when that component was last started, the IP addresses at which the component can be reached, and the version of the component. A given component type may have more than one row in this table if that component has been replicated for high availability or scale out.

View Domain Projects

Projects are groupings that allow designated users to view the same set of content resources within a region. Projects protect topology templates, infrastructure designs, and realized resources (VMs, volumes, etc., that have been created) from being visible to all users.

Domain Administrators are responsible for creating projects and securing them by role assignments.

To view existing domain projects

1. In the left navigation frame on the **Domain** tab, click **Projects**. Existing projects are listed alphabetically by Name.

2. If there are more than 20 projects, navigate using the link at the bottom of the list. See "Settings" on page 19 to change the table size.

3. To filter this list, click (🔍) and enter the first several characters of the Name or consecutive characters within the Name. Then click the **Filter All** button. To reset, click **Filter All** again. (Enter works the same as Filter All.)

See the following related topics:

- "Create a New Project" on next page

- "Add a User to a Project" on page 25

- "Add a Project Role to a User" on next page

- "Remove a Project Role from a User" on page 27

- "Remove a User from a Project" on page 27

- "Edit a Project" on page 27

- "Delete a Project" on page 28

- "View Project Usage Summary" on page 48

View Domain Project Usage Summary

1. In the left navigation frame of the Domain tab, click **Projects**.

2. Select the project and click [icon], **View Usage**. The Project Usage Summary displays the current month usage.

3. You can select a different month to view and click the **Submit** button.

4. The **Download CSV Summary** button opens a text file in Microsoft Excel with current month usage information.

Create a New Project

When creating a new project, the only required entry is Name. Users can be added at any time, and the Integrations and Limits settings have defaults settings.

For more information about domain projects, see "View Domain Projects" on previous page.

To create a new project

1. In the left navigation frame on the Domain tab, click Projects.

2. Click the **Create New Project** button.

3. On the **Project Info** tab, provide the following information:
   a. Name (required)

   b. Description (optional)

   c. Enabled - this option is checked by default, meaning project content is available to assigned users.

4. On the User tab, you can select a Project Administrator now or later.

5. On the Integrations tab, checking **CDA Application** makes this project available to HP Continuous Delivery Automation (HP CDA) and auto-creates users that are necessary for accessing from HP CDA. However, you still have to configure access to this project within HP CDA.

6. On the Limits tab, select the Region where the project should be located. **Absolute limits** represents the limit to the amount of system resources of the specified type that can be consumed at one time by users of the project.

7. Click the **Create Project** button to save. If the save was successful, a green status box displays in the upper right corner of the window and the **Enabled** field is set to True. If save was unsuccessful, a red status box displays an error message.

8. The new project displays with a system-assigned Project ID.

Add a User to a Project

You can add a user as a Project Administrator or Project User.

> **Note**: As an Administrator, if you add users to a project while they are logged in they may be logged out without warning or receive an error message. If so, users should sign out and sign back in. Carefully coordinate with users or schedule adding users to a project when they are not logged in.

To add a user to a project

1. In the left navigation frame on the **Domain** tab, click **Projects**.

2. Click the **Modify Users** button on the selected project.

3. In the Add New Users section of this window, select the user and then click the **Add To Project** button.

4. Select the Project Role and click the **Add User** button. The user is added to the Users For Project list.

Add a Project Role to a User

Once a user has been added to a project, you can add additional project roles to the user.

To add a project role to a user

1. In the left navigation frame of the **Domain** tab, click **Projects**.

2. Click the **Modify Users** button.

3. In the Users For Project list, select the user and click **Add Project Role**.

4. Select the project role and click **Add Role**.

5. Additional roles:

Role	Content Type	Place	Role Abilities
binding-designer	Binding Designer	Domain	Access Binding Designer

Role	Content Type	Place	Role Abilities
compute_admin	Nova: Instances, Volumes, Security Groups, Keypairs	Domain > Region > Project	All
	Default Security Group	Domain > Region	All
	Machine Flavor	Domain > Region	Creation
	Project	Domain > Region	Creation
	Project: Quota	Domain > Region > Project	Creation
	User/Role Association	Project	Visibility/Creation
	Network	Domain > Region	Visibility/Creation
	Network Project Association	Domain > Region > Project	Visibility/Creation
	Floating IP	Domain > Region	Creation
compute_user	Nova: Instances, Volumes, Security Groups, Keypairs	Domain > Region > Project	All
	Default Security Group	Domain > Region	Visibility
	Machine Flavor	Domain > Region	Visibility
	Project	Domain > Region	Visibility
	Project: Quota	Domain > Region > Project	Visibility
	Floating IP	Domain > Region	Visibility
	Floating IP Project Association	Domain > Region > Project	Creation
document_consumer	Focus:Topology Document	Domain > Project	Visibility
document_creator	Focus:Topology Document	Domain > Project	Creation
document_publisher	Visibility	Domain > Project	Publication
image_consumer	Glance: Image, Snapshots	Domain > Project	Visibility
image_creator	Glance: Image, Snapshots	Domain > Project	Creation
image_publisher	Glance: Image, Snapshots	Domain > Project	Publication

Role	Content Type	Place	Role Abilities
job_consumer	Eden: Job	Domain > Project	Visibility
rp_consumer	Peer: Resource Pool	Domain > Project	Visibility
rp_creator	Peer: Resource Pool	Domain > Project	Creation
service_map_ consumer	Eden: Service Map	Domain	Visibility
template_ designer	Template Designer	Domain	Access Topology Designer
topology_ creator	Eve: Realized Topology	Domain > Project	Creation

Remove a Project Role from a User

If only one role is listed for a user and you remove it, the user is removed from the **Users For Project** list and moved to the **Users** list.

To remove a project role from a user

1. In the left navigation frame of the Domain tab, click **Projects**.

2. Select the project and click the **Modify Users** button.

3. In the Users For Project list, select the user and click ▼ , **Remove Project Role**.

4. Select the role to remove and click the **Remove Role** button. That project role is no longer associated to the user.

Remove a User from a Project

To remove a user from a project

1. In the left navigation frame of the Domain tab, click **Projects**.

2. Select the project and click the **Modify Users** button.

3. In the **Users For Project** list, select the user and click ▼ , **Remove from Project**. The user is moved to the **Add New Users** list.

Edit a Project

Allows you to update project details and limits.

To edit a project

1. In the left navigation frame of the Domain tab, click **Projects**.

2. On the selected project, click ▼ , **Edit Project**.

3. On the Project Info tab, you can change the Name, Description, and whether the project is Enabled.

4. On the Limits tab, select a region and then edit the limits.

5. Click **Update Project**.

Delete a Project

Deleting a project marks the project for deletion with a ~ (tilde). The project is not completely deleted until a scheduled script runs to ensure there are no resources attached to the project. You have the option to restore the project any time before the script runs.

To delete a project

1. In the left navigation frame of the Domain tab, click **Projects**.

2. Select the project to delete and click ▼ , **Delete Project**. This option is red indicating the importance of this action.

3. You are prompted to confirm your decision. To confirm, click **Delete Project.**

4. The project list redisplays and the deleted project moves to the bottom of the list. The deleted project has a tilde (~) preceding the Project Name, indicating it will be permanently removed when a scheduled script runs to ensure there are no resources attached.

To restore a project

1. In the left navigation frame of the Domain tab, click **Projects**.

2. Select the project to restore, which will display at the bottom of the Projects list with a ~ preceding the Name.

3. Click ▼ , **Restore Project**. You are prompted to confirm your decision.

4. Click the **Restore** button.

5. The project is restored and listed alphabetically with no ~ preceding the Name.

View Domain Images

A VM is launched based on an image. Within the Cloud Administration Dashboard, Domain Images shows all non-snapshot images with the following formats:

- AMI (Amazon Machine Image)

- ARI (Amazon Ramdisk Image) - EC2 file

- AKI (Amazon Kernel Image) - EC2 file

- RAW

- QCOW2

Images are stored at the Domain level with a single Glance service and are unaffected by region selection. Only **Public** images are visible to users.

To view domain images

1. In the left navigation frame of the Domain tab, click **Images**.

2. If there are more than 20 images, navigate using the link at the bottom of the list. See to change the table size.

3. To filter this list, click (🔍) and enter the first several characters of the Name or consecutive characters within the Name. Then click the **Filter All** button. To reset, click **Filter All** again. (<Enter> works the same as **Filter All**.)

To view domain image details

Click the Name link in the **Image Name** column. The Image Overview window opens with read-only image information, specs, and custom properties.

Launch an Image

Only AMI, RAW, and QCOW2 format images can be launched. Launching provisions to the cloud.

To launch an image

1. In the left navigation frame of the **Domain** tab, click **Images**.

2. Select an image and click the **Launch** button.

3. The Project and User tabs populate with information from the selected image, so the launch window opens to the **Details** tab.

4. Enter the **Server Name** (required) and edit other details as needed:

   a. Server Name - name the host you are creating (required) **NOTE**: The instance can be launched at this point, if desired, with system defaults for remaining options.

   b. Flavor - Flavor indicates the size of the node. When you select a flavor here, the **Flavor Details** table on the left changes to indicate the node properties and values associated with your size selection.

   c. Instance Count - can be more than 1

5. Use the **Access & Security** tab to control access to your instance via keypairs and security groups.

   e. Select a keypair - Keypairs are ssh credentials which are injected into images when they are launched. Click (+) to import a keypair. Creating a new keypair here registers the public key and downloads the private key (a .pem file).

   f. Security Groups - default is checked, meaning users within this group can communicate on any port desired.

6. Use the **Volume Options** tab to select boot options for an instance launching with attached storage.

7. The **Post Creation** tab allows you to customize the instance by adding a script to run after launch of the instance. (The Customization Script field is analogous to "User Data" in other systems.)

8. Click **Launch**. If successful, the image is added to the Images list with an **Active** status.

Create a Domain Image

Allows you to create and configure an image to upload to the Image Service. Only images available via an HTTP URL are supported.

To create a domain image

1. In the left navigation frame of the **Domain** tab, click **Images**.

2. Click the **Create Image** button.

3. Enter the **Image Name**.

4. Enter the **Image Location**, which must meet the following criteria: (Compressed image binaries (.zip, .tar.gz) are supported.)

   a. A valid and direct HTTP URL to the image binary. URLs that redirect or serve error pages will result in unusable images.

   b. Image must be accessible to the Image Service.

5. Select a **Format**. NOTE: Only AMI, RAW, and QCOW2 images are launchable.

6. Enter **Minimum Disk** space and **Ram**.

7. Check **Public** if you want the image visible to all users.

8. Click the **Create Image** button. If successful, the image displays with an Active status.

Edit an Image

Allows you to modify the image name and whether it is public.

To edit an image

1. In the left navigation frame of the **Domain** tab, click **Images**

2. If the image is in AMI, RAW, or QCOW2 format, click [image] , **Edit Image**. For other image formats, click the **Edit Image** button.

3. Edit the **Name** and **Public** visibility as desired.

4. Click the **Update Image** button.

Delete an Image

Deleting an image may affect the resource pool.

To delete an image

1. In the left navigation frame of the **Domain** tab, click **Images**.

2. Select the image and click [image] , **Delete Image**. You are prompted to confirm your decision.

3. Click the **Delete Image** button. The Images list redisplays and the image has been removed.

To delete multiple images

1. Use the left checkbox(es) to select multiple images. To select all, click the top checkbox in the header row.

2. Click the **Delete Images** button.

3. You are prompted to confirm your decision.

View Domain Users

Users within a domain include both people and services.

To view domain users

1. In the left navigation frame of the Domain tab, click **Users**.

2. If there are more than 20 users, navigate using the link at the bottom of the list. See "Settings" on page 19 to change the table size.

3. To filter this list, click ([image]) and enter the first several characters of the Name or consecutive characters within the Name. Then click the **Filter All** button. To reset, click **Filter All** again. (<Enter> works the same as **Filter All**.)

To view user details

Click the Name link in the User Name column. The User Detail window opens with user and project information. If there is a Default Project listed, it was assigned during user creation. All projects the user is associated with display in the Project Information section.

To send an email to a user

Click the email link in the list of users. A new email is automatically created addressed to the user.

Create a New User

When creating a new user, a name, email, and password are required. You can optionally assign the user to one or more projects and assign them a project role and a domain-specific role during user creation.

To create a new user

1. In the left navigation frame of the Domain tab, click **Users**.

2. Click the **Create User** button. The Create User window opens.

3. On the User tab, enter the User Name, Email, and a Password. All fields are required.

4. Click the Project tab if you want to assign the user to a project.

5. Select a project from the list. The project selected here becomes the Default Project for the user. Assign the user a role as either a Project Administrator or Project User. **NOTE**: If you don't add the user to a project in this step, you can add them through the Projects tab. See "Add a User to a Project" on page 25

6. Click the Domain tab if you want to assign the user domain-specific role(s):
   a. Domain Administrator - all administrator rights plus Cloud Connector

   b. Domain Architect - ability to set up images, flavors, and topology components

6. Click **Create User**. If the save was successful, a green status box displays in the upper right corner of the window and the **Enabled** field is set to True. If save was unsuccessful, a red status box displays an error message.

To view user details

Click the name in the User Name column. The User Detail window opens with user and project information. If there is a Default Project listed, it was assigned during user creation. All projects the user is associated with displays in Project Information.

To send an email to a user

Click the email link in the list of users. A new email is automatically created addressed to the user.

Update a User

To update user information

1. In the left navigation frame of the Domain tab, click **Users**.

2. Select the user from the list and click the **Edit User** button.

3. Update the Name, Email, or Password on the User tab.

4. Click the Domain tab to update the domain role of the user.

5. Click **Update User** to save. If the save was successful, a green status box displays in the upper right corner of the window and the **Enabled** field is set to True. If save was unsuccessful, a red status box displays an error message.

Disable/Enable a User

A disabled user remains visible in the Users list with an **Enabled** status of False.

To disable a user

1. In the left navigation frame of the **Domain** tab, click **Users**.

2. Select the user and click ▼, **Disable User**.

3. The user remains on the list but the Enabled field changes to False.

To enable a user

1. In the left navigation frame of the **Domain** tab, click **Users**.

2. Select the user and click ▼, **Enable User**.

3. The user remains on the list and the Enabled field changes to True.

Delete a User

Any user name or type can be deleted except the Admin / Domain Administrator.

To delete a user

1. In the left navigation frame of the **Domain** tab, click **Users**.

2. Select the user to delete and click ▼, **Delete Project**. This option is red indicating the importance of this action.

3. You are prompted to confirm your decision. To confirm, click **Delete User.**

4. The Users list redisplays and the deleted user moves to the bottom of the list. The tilde (~) preceding the **User Name** indicating it will be permanently removed when a scheduled script runs to ensure there are no resources attached.

To restore a deleted user

Deleting a user marks the user for deletion with a ~ (tilde) and moves them to the bottom of the list. The user is not completely deleted until a scheduled script runs to ensure there are no resources attached. You have the option to restore the user any time before the script runs.

1. Select the user and click the **Restore User** button.

2. Click **Restore**. The user is added back into the Users list alphabetically.

View Domain Roles

You can view role name and ID, and add and delete user roles from this window.

To view domain roles

1. In the left navigation frame on the Domain tab, click **Roles**. Existing roles are listed alphabetically by Name.

2. If there are more than 20 roles, navigate using the link at the bottom of the list. See "Settings" on page 19 to change the table size.

3. To filter this list, click (🔍) and enter the first several characters of the Name or consecutive characters within the Name. Then click the **Filter All** button. To reset, click **Filter All** again. (<Enter> works the same as **Filter All**.)

See the following related topics:

"Create a Role" below

"Delete a Role" on next page

Create a Role

When creating roles in Cloud Connector, the authorization code checks if a service requires the specified role and whether the requesting user has the role. The service defines which roles are required. If you are using Keystone for your own purposes or with other Cloud Connector services, you may want to define your own roles.

To create a role

1. In the left navigation frame of the **Domain** tab, click **Roles**.

2. Click the **Create Role** button.

3. Enter the Role Name and click the **Create Role** button.

4. The role is added to the list alphabetically and assigned a Role ID.

Delete a Role

The only roles that cannot be deleted are:

- admin

- arch

- proj_admin

- project_user

Once deleted, a role is removed throughout the system.

To delete a role

1. In the left navigation frame of the **Domain** tab, click **Roles**.

2. Select the role to delete and click the **Delete Role** button.

3. You are prompted to confirm your decision, as deleting a role cannot be undone. To confirm, click **Delete Role.**

4. The role is removed from the list and throughout the system.

To delete multiple roles

1. Use the left checkbox(es) to select multiple roles.

2. Click the **Delete Roles** button.

3. You are prompted to confirm your decision. The roles are removed from the list and throughout the system.

Regions

A **Region** is a compute grouping that provides context for the usage of compute servers, networking systems, and storage resources. A Region makes use of a defined set of users, projects, topology templates, infrastructure designs, and services governed by a single Cloud Connector Keystone domain service. Regions are used to group different locations, different types of hardware, or to separate different resources belonging to different groups.

Region services are registered under the names:

- Nova - A region is constructed by installing and naming the Cloud Connector compute service called Nova.

- Volume - the creation and management of volumes is controlled by the Volume service which is accessed through the Nova service interface.

- EC2 - An alternate service interface to the Nova service that is compatible with the Amazon EC2 service.

Region services can be run on the same node as the domain controller or they can be run on a different server. There can be multiple regions within a domain.

Visibility by role

Within the Region tab, visibility is by role. Administrators see all tabs, users only two:

- Overview

- Services

- Instances

- Volumes

- Networks

- Flavors - Administrators and Users

- Limits - Administrators and Users

To select a region

If there is more than one region defined for your domain, you select the region to manage by clicking **Current Compute Region** in the left navigation frame of the **Region** tab. **NOTE**: Changing regions may switch the point of action to the **Project** tab. Re-click the **Region** tab and the new selected region displays.

View Region Usage Summary

1. In the left navigation frame of the Region tab, click **Overview**.

2. The Region Usage Summary displays the current month usage.

3. You can select a different month to view and click the **Submit** button.

4. The **Download CSV Summary** button opens a text file in Microsoft Excel with current month usage information.

View Region Services

Region Services displays a read-only list of services that are registered in the region.

To view region services

1. In the left navigation frame of the Region tab, click **Services**. The following region services are listed:

Service Name	Description
ec2	An alternate service interface for the Nova service.
nova	A region is constructed by installing and naming a compute server called Nova.
volume	Manages attached storage for the VMs.

2. The **Service URL (Public)** column shows the IP address where the service is running.

3. **Status** - Enabled

View Region Instances

Instances are all of the VMs running within the region.

To view region instances

1. In the left navigation frame of the **Region** tab, select the **Current Compute Region** you want to view.

2. Select **Instances**. This action displays the state of the instances running in the region.

Launch Region Instance

The system allows you to launch an instance only into a project of which you are a member. Even as a Domain Administrator, if you are not a member of a project you cannot launch an instance into the project, even though you can create compute resources in the project.

Make your selections carefully when setting up an Instance, as the only editable property once it is launched is the **Name**.

To launch a region instance

1. In the left navigation frame of the Region tab, click **Instances**.

2. Click the **Launch Instance** button.

3. Optionally select the Project and User for whom the instance should be created.

4. Click the **Details** tab to specify the details for launching an instance.
   a. **Instance Source** - an Image (default) or Snapshot

   b. **Image** - depending on your selection in Instance Source, you are required to select an AMI, RAW, or QCOW2 formatted image to create the new VM or a specific snapshot.

   c. **Server Name** - name the host you are creating (required) **NOTE**: The instance can be launched at this point, if desired, with system defaults for remaining options.

   d. **Flavor** - Flavor indicates the size of the node. When you select a flavor here, the **Flavor Details** table on the left changes to indicate the node properties and values associated with your size selection.
      i. Name - tiny to xlarge or the name of a custom flavor that has been created

      ii. Number of virtual CPUs

      iii. Root Disk - the one that gets assigned by the image

      iv. Ephemeral Disk - a second storage disk available to the VM

      v. RAM - processor memory

      vi. Instance Count - can be more than 1

5. Use the **Access & Security** tab to control access to your instance via keypairs and security groups.
   e. Select a keypair - Keypairs are ssh credentials which are injected into images when they are launched. Click (+) to import a keypair. Creating a new keypair here registers the public key and downloads the private key (a .pem file).

   f. Security Groups - Security groups are one way to provide a measure of network isolation between different VMs, as they limit the ports into which network traffic can enter the group. You can select multiple groups.

6. Use the **Volume Options** tab to select boot options for an instance launching with attached storage.

7. The **Post Creation** tab allows you to customize the instance by adding a script to run after launch of the instance. (The Customization Script field is analogous to "User Data" in other systems.)

8. Click **Launch**. The Instance is added to the Instances list with a **Power State** of **Running**.

View Region Instance Details

The Instance Overview is a read-only summary of all properties related to the Instance.

To view instance details

1. In the left navigation frame of the Region tab, click **Instances**

2. Click the link on the **Instance Name**.

3. The Instance Overview displays with the following information:
   a. Name

   b. ID

   c. Status

   d. Flavor

   e. IP Addresses

   f. Security Groups

   g. Metadata

   h. Volumes Attached

View the Region Instance Console Log

The Instance Console Log is the system log from the VM and is used for debugging and troubleshooting. You can view the full instance log or specify the number of lines of the log file to view.

To view the instance console log

1. In the left navigation frame of the Region tab, click **Instances**.

2. Select the instance and click [icon], **View Log** (or access the Console Log by clicking the linked **Instance Name**).

3. Click the **Log** tab to view the most recent 35 lines from the Instance Console Log.

4. To change the number of display lines, enter a value in **Log Length** and click the **Go** button.

5. Click the **View Full Log** button to view all logged entries.

Using the Instance VNC Console

This option allows you to remotely log into the VM directly.

To access the VNC Console

1. In the left navigation frame of the Region tab, click **Instances**.

2. Select the instance and click [icon], **VNC Console** or access the VNC Console by clicking the linked **Instance Name.**

Edit an Instance

The only Instance property you can edit is the Name.

To edit an instance

1. In the left navigation frame of the Region tab, click **Instances**

2. Select the Instance and click on the **Edit Instance** button.

3. Update the **Name** and click on the **Save Changes** button. The list redisplays with the new name.

Create an Instance Snapshot

Once you have set up, installed, and configured an instance, you can create a snapshot to preserve the disk state of the running instance. Snapshots can then be used to start up another VM in the future.

To create a snapshot

1. In the left navigation frame of the **Region** tab, click **Instances**.

2. Select the instance and click ▾, **Create Snapshot**.

3. The Instance ID defaults to the current instance and cannot be changed.

4. Enter a name for the snapshot.

5. Click the **Create Snapshot** button.

6. The Snapshots window opens in the Project tab displaying Instance Snapshots alphabetically in the top report (Volume Snapshots at the bottom).

Pause an Instance

Pausing an instance stores the content of the VM in memory (RAM). This keeps the instance running but in a frozen state. A paused instance could be compared to a PC in hibernation or sleep mode.

To free up memory and vCPUs, you should suspend an instance. See "Suspend an Instance" below.

To pause an instance

1. In the left navigation frame of the Region tab, click **Instances**.

2. Select the instance and click ▾, **Pause Instance**.

3. The Status and Power State of the instance show **Paused**.

To unpause an instance

1. Select the instance and click ▾, **Unpause Instance**.

2. After a short time, the Status returns to Active and the Power State returns to Running.

Suspend an Instance

Suspending an instance frees up memory and vCPUs. Suspension could be compared to hibernation mode.

To suspend an instance

1. In the left navigation frame of the **Region** tab, click **Instances**.

2. Select the instance and click [▼] , **Suspend Instance**.

3. The Task column indicates the instance is suspending. When complete, the Status shows Suspended and the Power State shows Shutoff.

To resume an Instance

1. Select the instance and click [▼] , **Resume Instance**.

2. The Task column indicates the instance is resuming. When complete, the Status returns to Active and the Power State returns to Running.

Reboot an Instance

During a reboot, the VM and all the services it hosts are not available to users.

To reboot an instance

1. In the left navigation frame of the Region tab, click **Instances**.

2. Select the instance and click [▼] , **Reboot Instance**.

3. You are prompted to confirm this action. Click the **Reboot Instance** button to continue.

4. The Task column indicates the instance is Rebooting Hard. If reboot was successful, the Status shows Active again and the Power State shows Running.

Terminate Region Instance

When you terminate an instance, the VM is shut down and the resources associated with it are released. It is not possible to start the VM after it is terminated because it no longer exists. Termination is scheduled because it takes some time to free all the resources.

When an instance is terminated:

- Any changes associated with the volume that the image is running on are discarded.

- Any changes associated with the volume implied by the disk space (ephemeral) that are specified in the flavor are discarded.

- Any changes associated with attached volumes are retained in the attached volume and could be reattached to a different VM.

- The VM is removed from any security group it was associated with, and frees its floating IP addresses.

To terminate an instance

1. In the left navigation frame of the **Region** tab, click **Instances**.

2. Select the instance and click ▼ , **Terminate Instance**. This option is red indicating the importance of the action.

3. You are prompted to confirm this action. Click the **Terminate Instance** button to continue.

4. The **Task** column indicates the instance is Deleting. If terminate was successful, the green Status box in the upper right indicating the system has scheduled the termination of the instance.

To terminate multiple instances

1. Use the left checkbox(es) to select multiple instances. To select all, click the top checkbox in the header row.

2. Click the **Terminate Instances** button.

3. You are prompted to confirm your decision. Click the **Terminate Instances** button to continue.

View Region Volumes

Volumes are the disk storage volumes that have been created in the region. Volumes are persistent, meaning you can create data in these volumes and attach to various VMs (instances) over time as needed.

To view region volumes

In the left navigation frame of the **Region** tab, click **Volumes**. This list displays the existing volumes available in the region.

View Region Volume Detail

To view region volume details

1. In the left navigation frame of the **Region** tab, click **Volumes**. This list displays the existing volumes available in the region.

2. Click the linked **Volume Name** to view the Volume Overview.

Create a New Volume

Volumes can be created manually by entering the size, or you can use a volume snapshot as a source.

After a volume has been created it is available to attach to VMs. Attaching the volume to a VM allows content to be put onto the volume. Then it can be detached (with content) and be made available to be attached to other VMs.

To create a new volume

1. In the left navigation frame of the **Region** tab, click **Volumes**.

2. Click the **Create Volume** button. The **Volume Quotas** show you how much disk space is available and the number of volumes available to create.

3. If you are using a snapshot as a source for the new volume, select the snapshot from the field drop-down and it auto-populates the **Volume Name** and **Size**. Enter an optional **Description**.

4. If you are creating a new volume not based on a snapshot, enter the **Volume Name** and **Size**. **Description** is optional.

5. Click the **Create Volume** button.

Attach a Volume to an Instance

Volumes are disk storage volumes that have been created in a project and can be attached to an Instance. A volume can only be attached to one VM at a time. While attached, that VM has sole use of the volume and the volume cannot be attached to another VM.

To attach a volume to an instance

1. In the left navigation frame of the Region tab, click **Volumes**.

2. Select the volume and click ▼ , **Attach Volume**.

3. The Current Attachments are listed at the top; you attach an instance by making a selection in the **Instance** field.

4. As the volume attaches, the status changes. In-use status indicates a volume is in use by a VM.

To detach a Volume

Once a volume is attached, the **Detach Volume** button is available through the **Attach Volume** action.

Create a Volume Snapshot

Once you have set up a volume, you can create a snapshot to preserve the current state of the volume. Volume Snapshots can then be used to start up another VM in the future.

To create a volume snapshot

1. In the left navigation frame of the **Region** tab, click **Volumes**.

2. Select the volume and click ▼, **Create Snapshot**.

3. Enter a Name for the snapshot and optionally a Description.

4. Click the **Create Snapshot** button.

5. The system switches to **Project > Images** and the status box in the upper right indicates snapshot creation progress.

Delete a Volume

Deleting a volume means the volume will no longer exist, the data will be lost, it cannot be attached to any VM, and no user can access. Deleting a volume cannot be undone.

To delete a volume

1. In the left navigation frame of the Region tab, click **Volumes**.

2. Select the volume and click ▼, **Delete Volume**.

To delete multiple volumes

1. Use the left checkbox(es) to select multiple volumes. To select all, click the top checkbox in the header row.

2. Click the **Delete Volumes** button.

3. You are prompted to confirm your decision.

View Region Networks

The Region **Networks** tab is only viewable to an Administrator if Nova and Cloud Administration Dashboard are on the same server.

To view region networks

In the left navigation frame of the Region tab, click **Networks**.

- The **Fixed IP Networks** section identifies private services and how networking was configured when the servers were designated for use within the cloud. Fixed IP Networks are configured with Cloud Installation Dashboard and typically won't be modified unless the physical network configuration changes.

- The **Floating IP Network** section lists a set of public addresses that are available for associating to VMs that need to be visible outside the cloud but within the enterprise. This area outside the cloud but within an enterprise is considered the Cloud Edge.

Create a Fixed IP Network

Creating a Fixed IP Network allows you create a virtual network definitions within the cloud and assign it to a project. Addresses are visible only inside the cloud.

To create a fixed IP network

1. In the left navigation frame of the **Region** tab, click **Networks**.

2. Click the **Create Fixed IP Network** button.

3. Enter the following information.

Option	Description
Network Label	Label for the network
IPv4 CIDR	IPv4 subnet for fixed IPs. The IPv4 CIDR option is a CIDR block which describes the IP address space for all of the instances. This space will be divided up into subnets. This range is typically a private network.
IPv6 CIDR	Fixed Range V6
Gateway IPv4	Gateway Version 4
Gateway IPv6	Gateway Version 6
DNS 1	First DNS
DNS 2	Second DNS
Vlan ID	Vlan ID
Multi-host	True or False

Option	Description
Number of Networks	Number of networks to create
Network Size	Number of IPs per network
Bridge ID	VIFs on this network are connected to this bridge
Bridge Interface	The bridge is connected to this interface
Priority	Network interface priority

4. Click the **Create Network** button.

Create a Floating IP Pool

You associate floating IP addresses with VMs that need to be visible outside the cloud but within the enterprise. For example, a VM that is to be used as a web server might need to be accessed from outside the cloud and should get assigned a floating IP address.

To create a floating IP network

1. In the left navigation frame of the **Region** tab, click **Networks**.

2. Click the **Create Floating IP** button.

3. Enter field information.

4. Click the **Create Network** button.

View Region Flavors

Flavors are machine configurations that can be used to create new VMs. A flavor describes the amount of memory, number of CPUs, and ephemeral disk space available to the VM.

To view region flavors

In the left navigation frame of the Region tab, click **Flavors**. This list displays current flavors running in the region.

Create a Flavor

Creating a flavor defines a new configuration used to create new VMs. Typically, the pre-defined Flavors m1.tiny through m1.xlarge are sufficient.

To create a flavor

1. In the left navigation frame of the **Region** tab, click **Flavors**.

2. Click the **Create Flavor** button.

3. All fields are required. All fields are integer values except for the **Name** field.

4. Click the **Create Flavor** button. The Flavors list redisplays with the new flavor list alphabetically and a system-assigned Flavor ID.

Delete Flavors

Deleting a flavor has no effect on existing VMs that were created using that flavor. Users will not be able to specify a flavor that has been deleted when they create new VMs.

NOTE: If all flavors are deleted, you will not be able to create VMs.

To delete a flavor

1. In the left navigation frame of the Region tab, click **Flavors**.

2. Select the flavor to delete and click the **Delete Flavor** button in that row.

3. You are prompted to confirm your decision, as deleting a flavor cannot be undone. To confirm, click **Delete Flavor.** The flavor is removed from the list.

To delete multiple flavors

1. Use the left checkbox(es) to select multiple flavors. To select all, click the top checkbox in the header row.

2. Click the **Delete Flavors** button.

3. You are prompted to confirm your decision.

View Region Limits

Used to view the limits for the various region resources.

Absolute Limits represent the maximum amount of system resources of the specified type that can be consumed at one time by users of the project. Limits are set up when you create or edit a project in the Domain tab. The limits here are defaults used when a project is created.

Default Rate Limits shows limits associated with a unit of time. For example, a user may be limited to sending no more than 30 HTTP requests per minute.

Projects

 Projects are groupings that allow users to view the same set of resources. Projects are the unit of ownership within the Cloud Connector and protect topology templates, infrastructure designs, and

regional resources from being visible to all users.

The Project tab and related tasks are available to both administrators and users of the Cloud Connector system, including launching Topology Designer.

If you have access to multiple projects, select from the **Current Project** list in the left navigation frame. Once a project is selected, you can

- view project usage and manage instances, volumes, images, snapshots, and security
- view topologies
- create and manage infrastructure topology templates and resource binding documents that are used for provisioning
- create and manage resource pools

View Project Usage Summary

To view project usage

1. In the left navigation frame of the Project tab, select the **Current Project** you want to view.
2. Click **Overview**. The Project Usage Summary displays the current month usage.
3. You can select a different month to view and click the **Submit** button.
4. The **Download CSV Summary** button opens a text file in Microsoft Excel with current month usage information.

View Project Instances

Instances are all of the VMs running within the selected project.

To view project instances

1. In the left navigation frame of the **Project** tab, select the **Current Project** you want to view.
2. Select **Instances**. This action displays the state of the instances running in the project.

Launch an Instance within a Project

The system allows you to launch an instance only into a project of which you are a member. Even as a Domain Administrator, if you are not a member of a project you cannot launch an instance into the project, even though you can create compute resources in the project.

Make your selections carefully when setting up an Instance, as the only editable property once it is launched is the **Name**.

To launch an instance

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**

2. Click **Instances**.

3. Click the **Launch Instance** button. This action opens the **Details** tab, as the Project & User tab was auto-populated with current information.

4. Specify the details for launching an instance.
   a. **Instance Source** - an Image (default) or Snapshot

   b. **Image** - depending on your selection in the **Instance Source** field, you are required to select an AMI, QCOW2, or RAW formatted image or a specific snapshot. (ARI and AKI types cannot be launched.)

   c. **Server Name** - name the host you are creating (required) **NOTE**: The instance can be launched at this point, if desired, with system defaults for remaining options.

   d. **Flavor** - Flavor indicates the size of the node. When you select a flavor here, the **Flavor Details** table on the left changes to indicate the node properties and values associated with your size selection.
      i. Name - tiny to xlarge or the name of a custom flavor that has been created

      ii. Number of virtual CPUs

      iii. Root Disk - the one that gets assigned by the image

      iv. Ephemeral Disk - a second storage disk available to the VM

      v. RAM - processor memory

      vi. Instance Count - can be more than 1

5. Use the **Access & Security** tab to control access to your instance via keypairs and security groups.
   e. Select a keypair - Keypairs are ssh credentials which are injected into images when they are launched. Click (+) to import a keypair. Creating a new keypair here registers the public key and downloads the private key (a .pem file).

   f. Security Groups - Security groups are one way to provide a measure of network isolation between different VMs, as they limit the ports into which network traffic can enter the group. You can select multiple groups.

6. Use the **Volume Options** tab to select boot options for an instance launching with attached storage.

7. The **Post Creation** tab is analogous to "User Data" in other systems. This tab allows you to customize the instance by adding a script to run after launch of the instance.

8. Click **Launch**. The Instance is added to the Instances list with a **Power State** of **Running**.

View Project Instance Details

The Instance Overview is a read-only summary of all properties related to the Instance.

To view instance details

1. In the left navigation frame of the Project tab, click **Instances**.

2. Click the link on the **Instance Name**.

3. The Instance Overview displays with the following information:
   a. Name

   b. ID

   c. Status

   d. Flavor

   e. IP Addresses

   f. Security Groups

   g. Metadata

   h. Volumes Attached

View the Project Instance Console Log

The Instance Console Log is the system log from the VM and is used for debugging and troubleshooting. You can view the full instance log or specify the number of lines of the log file to view.

To view the instance console log

1. In the left navigation frame of the Project tab, click **Instances**.

2. Select the instance and click [▼], **View Log** (or access the Console Log by clicking the linked **Instance Name**).

3. Click the **Log** tab to view the most recent 35 lines from the Instance Console Log.

4. To change the number of display lines, enter a value in **Log Length** and click the **Go** button.

5. Click the **View Full Log** button to view all logged entries.

Using the Project Instance VNC Console

This option allows you to remotely log into the VM directly.

To access the VNC Console

1. In the left navigation frame of the Project tab, click **Instances**.

2. Select the instance and click ▼ , **VNC Console** or access the VNC Console by clicking the linked **Instance Name**.

Edit an Instance

The only Instance property you can edit is the Name.

To edit an instance

1. In the left navigation frame of the Project tab, click **Instances**.

2. Select the instance and click ▼ , **Edit Instance**.

3. Update the **Name** and click on the **Save Changes** button. The list redisplays with the new name.

Create an Instance Snapshot

Once you have set up, installed, and configured an instance, you can create a snapshot to preserve the disk state of the running instance. Snapshots can then be used to start up another VM in the future.

To create a snapshot

1. In the left navigation frame of the **Project** tab, click **Instances**.

2. Select the instance and click the **Create Snapshot** button.

3. The Instance ID defaults to the current instance and cannot be changed.

4. Enter a **Name** for the snapshot.

5. Click the **Create Snapshot** button.

6. The Snapshots list redisplays with the new snapshot inserted alphabetically in the top section of the report (Volume Snapshots display at the bottom).

Pause a Project Instance

Pausing keeps an instance running but in a "frozen" state and stores the content of the VM in RAM.

To pause an instance

1. In the left navigation frame of the Project tab, click **Instances**.

2. Select the instance and click ⏷ , **Pause Instance**.

3. The Status and Power State of the instance show **Paused**.

To Unpause an Instance

1. Select the instance and click ⏷ , **Unpause Instance**.

2. After a short time, the Status returns to Active and the Power State returns to Running.

Suspend an Instance

Suspending an instance frees up memory and vCPUs. Suspension could be compared to hibernation mode.

To suspend an instance

1. In the left navigation frame of the **Project** tab, click **Instances**.

2. Select the instance and click ⏷ , **Suspend Instance**.

3. The Task column indicates the instance is suspending. When complete, the Status shows Suspended and the Power State shows Shutoff.

To resume an Instance

1. Select the instance and click ⏷ , **Resume Instance**.

2. The Task column indicates the instance is resuming. When complete, the Status returns to Active and the Power State returns to Running.

Reboot an Instance

Rebooting an Instance is similar to cycling power on a physical server; the operating system is restarted from scratch. Rebooting temporarily makes the VM and all the services it hosts unavailable.

To reboot an instance

1. In the left navigation frame of the **Project** tab, click **Instances**.

2. Select the instance and click , **Reboot Instance**.

3. You are prompted to confirm this action. Click the **Reboot Instance** button to continue.

4. The **Task** column indicates the instance is Rebooting Hard. If reboot was successful, the **Status** shows Active again and the **Power State** shows Running.

Associate a Floating IP to an Instance

Allows you to associate a Floating IP address to an instance that needs to be visible outside the cloud.

To associate a floating IP to an instance

1. In the left navigation frame of the **Project** tab, click **Instances.**

2. Select the instance and click , **Associate Floating IP**.

3. Select an **IP Address** if there is one available. If not available, see Step 6.

4. Select an **Instance**.

5. Click the **Associate** button.

6. Click (+) to allocate a floating IP from a pool.

7. Select a **Pool** and then click the **Allocate IP** button.

Terminate Project Instance

When you terminate an instance, the VM is shut down and the resources associated with it are released. It is not possible to start the VM after it is terminated because it no longer exists. Termination is scheduled because it takes some time to free all the resources.

When an instance is terminated:

- Any changes associated with the volume that the image is running on are discarded.

- Any changes associated with the volume implied by the disk space (ephemeral) that are specified in the flavor are discarded.

- Any changes associated with attached volumes are retained in the attached volume and could be reattached to a different VM.

- The VM is removed from any security group it was associated with, and frees its floating IP addresses.

To terminate an instance

1. In the left navigation frame of the Project tab, click **Instances**.

2. Select the instance and click ⏷, **Terminate Instance**. This option is red indicating the importance of the action.

3. You are prompted to confirm this action. Click the **Terminate Instance** button to continue.

4. The Task column indicates the instance is Deleting. If terminate was successful, the green Status box in the upper right indicating the system has scheduled the termination of the instance.

To terminate multiple instances

1. Use the left checkbox(es) to select multiple instances. To select all, click the top checkbox in the header row.

2. Click the **Terminate Instances** button.

3. You are prompted to confirm your decision. Click the **Terminate Instances** button to continue.

View Project Volumes

Volumes are the disk storage volumes that have been created in the project. Volumes are persistent, meaning you can create data in these volumes and attach to various VMs (instances) over time as needed.

To view project volumes

In the left navigation frame of the **Project** tab, click **Volumes**. This list displays the existing volumes in the region and their **Status**, e.g., Attaching, Available, In-Use.

View Project Volume Detail

To view project volume details

1. In the left navigation frame of the Project tab, click **Volumes**.This list displays the existing volumes available in the project.

2. Click the linked **Volume Name** to view the Volume Overview.

Create a New Volume

Volumes can be created manually by entering the size, or you can use a volume snapshot as a source. Once the volume has been created, it is attached to a VM where content can be placed.

To create a new volume

1. In the left navigation frame of the **Project** tab, click **Volumes**.

2. Click the **Create Volume** button. The **Volume Quotas** shows:
   a. available disk space

   b. the number of volumes allocated

   c. the number of volumes available

Note: If you are logged in as an Administrator, Nova returns all volumes. When you attempt to create a new volume through the dashboard, you may see in **Volume Quotas** there are no volumes available to create. In this case, log out and log back in as a User rather than an Administrator.

3. If you are using a snapshot as a source for the new volume, select the snapshot from the field drop-down and it auto-populates the **Volume Name** and **Size**. Enter an optional **Description**.

4. If you are creating a new volume not based on a snapshot, enter the **Volume Name** and **Size**. **Description** is optional.

5. Click the **Create Volume** button.

6. The volume has been created and is available to attach to VMs.

Attach a Volume to an Instance

Volumes are disk storage volumes that have been created in a project and can be attached to an Instance. A volume can only be attached to a single VM at a time, and that VM has sole use of that volume.

To attach a volume to an instance

1. In the left navigation frame of the **Project** tab, click **Volumes**.

2. Select the volume and click ▾, **Attach Volume**. Attaching is a process that takes time, so the **Status** will vary.

3. The Current Attachments are listed at the top; you attach an instance by making a selection in the Instance field.

To detach a Volume

Once a volume is attached, the **Detach Volume** button is available through the Attach Volume action.

Create a Volume

Volumes are disk storage volumes that can be created in a project and can be attached to an Instance.

To create a volume

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Volumes**.

3. Click the **Create Volume** button.

4. Enter a **Name** for the snapshot and optionally a **Description**.

5. Click the **Create Volume** button. The new volume is added to the list alphabetically.

Delete a Volume

Deleting a volume means the volume will no longer exist, the data will be lost, it cannot be attached to any VM, and no user can access. Deleting a volume cannot be undone.

To delete a volume

1. In the left navigation frame of the **Project** tab, click **Volumes**.

2. Select the volume and click [▼], **Delete Volume**.

To delete multiple volumes

1. Use the left checkbox(es) to select multiple volumes. To select all, click the top checkbox in the header row.

2. Click the **Delete Volumes** button.

3. You are prompted to confirm your decision.

View Project Images

Project Images is a view of only the launchable images (AMI, QCOW2, and RAW format) that can be used to create new VMs. Images visible here are determined by the "Regions" on page 35.

To view project images

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Images** to display a list of existing images in the project.

To view image details

Click the linked **Image Name** to view general information, specs, and custom properties for the image.

Create an Image

Allows you to create and configure an image to upload to the Image Service. Only images available via an HTTP URL are supported. Compressed image binaries are supported (.zip, .tar.gz)

To create an image

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Images**.

3. Click the **Create Image** button.

4. Enter a **Name**.

5. Enter the **Image Location**. This must be a valid and direct URL to the image binary. URLs that redirect or serve error pages will result in unusable images.

6. Select a **Format**. NOTE: Only AMI, QCOW2, and RAW images are launchable.

7. Enter **Minimum Disk** and **Ram** space.

8. Check **Public** if you want image visible to all users.

9. Click **Create Image**. If successful, the image displays in the list as **Active**.

Launch an Image

Only AMI, QCOW2, and RAW format images can be launched. Launching provisions to the cloud.

To launch an image

1. In the left navigation frame of the **Domain** tab, click **Images**.

2. Select an image and click the **Launch** button.

3. The Project and User tab populate with information from the selected image, so the launch window opens to the **Details** tab.

4. Enter the **Server Name** (required) and edit other details as needed:

   a. Server Name - name the host you are creating (required) **NOTE**: The instance can be launched at this point, if desired, with system defaults for remaining options.

   b. Flavor - Flavor indicates the size of the node. When you select a flavor here, the **Flavor Details** table on the left changes to indicate the node properties and values associated with your size selection.

   c. Instance Count - can be more than 1

5. Use the **Access & Security** tab to control access to your instance via keypairs and security groups.

   e. Select a keypair - Keypairs are ssh credentials which are injected into images when they are launched. Click (+) to import a keypair. Creating a new keypair here registers the public key and downloads the private key (a .pem file).

   f. Security Groups - Security groups are one way to provide a measure of network isolation between different VMs, as they limit the ports into which network traffic can enter the group. Default is checked.

6. Use the **Volume Options** tab to select boot options for an instance launching with attached storage.

7. The **Post Creation** tab allows you to customize the instance by adding a script to run after launch of the instance. (The Customization Script field is analogous to "User Data" in other systems.)

8. Click **Launch**. If successful, the image is added to the Images list with an **Active** status.

Edit an Image

Allows you to modify the image name and whether it is public.

To edit an image

1. In the left navigation frame of the **Projects** tab, click **Images**.

2. If the image is in AMI format, click ⏷, **Edit Image**. For other image formats, click the **Edit Image** button.

3. Edit the **Name** and **Public** visibility as desired.

4. Click the **Update Image** button.

Delete an Image

Once an image is deleted, it is no longer possible to create instances based upon that image.

To delete an image

1. In the left navigation frame of the **Projects** tab, click **Images**.

2. Select the image and click ▼ , **Delete Image**. You are prompted to confirm your decision.

3. Click the **Delete Image** button. The Images list redisplays and the image has been removed.

To delete multiple images

1. Use the left checkbox(es) to select multiple images. To select all, click the top checkbox in the header row.

2. Click the **Delete Images** button.

3. You are prompted to confirm your decision.

View Project Snapshots

Snapshots preserve the disk state of running Instances and Volumes. Use the steps below to view snapshots that were created in the Instances section of the **Region** tab.

To view project snapshots

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Snapshots** to display a list of existing snapshots in the project.

To view project snapshot details

Click the linked **Image Name** in the Instance Snapshot list to view image details.

Launch an Instance Snapshot

To launch an instance snapshot

1. In the left navigation frame of the **Projects** tab, click **Snapshots**.

2. Select snapshot and click the **Launch** button.

3. The Project and User tabs populate with information from the selected image, so the launch window opens to the **Details** tab.

4. Select the **Instance Source**, either Image or Snapshot. The selection in this field determines what is available in the next field.

5. Select an **Instance Snapshot** or an **Image**.

6. Enter the **Server Name** (required) and edit other details as needed:

   - Server Name - name the host you are creating (required) **NOTE**: The instance can be launched at this point, if desired, with system defaults for remaining options.

   - Flavor - Flavor indicates the size of the node. When you select a flavor here, the **Flavor Details** table on the left changes to indicate the node properties and values associated with your size selection.

   - Instance Count - can be more than 1

7. Use the **Access & Security** tab to control access to your instance via keypairs and security groups.
   - Select a keypair - Keypairs are ssh credentials which are injected into images when they are launched. Click (+) to import a keypair. Creating a new keypair here registers the public key and downloads the private key (a .pem file).

   - Security Groups - a list of all security groups that have been created for this project. Security groups are one way to provide a measure of network isolation between different VMs, as they limit the ports into which network traffic can enter the group. The Default group is checked, but you can add multiple Security Groups.

8. Use the **Volume Options** tab to select boot options for the instance:
   - Don't boot from a volume.

   - Boot from a volume.

   - Boot from volume snapshot (creates a new volume).

9. The **Post Creation** tab allows you to customize the instance by adding a script to run after launch of the instance. (The Customization Script field is analogous to "User Data" in other systems.)

10. Click **Launch**. If successful, the image is added to the Instance Snapshots list with an **Active** status.

Edit an Instance Snapshot

Allows you to modify the Snapshot Name and whether it is public.

To edit an instance snapshot

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Snapshots** to display a list of existing snapshots in the project.

3. Select the snapshot to edit and click , **Edit Snapshot**.

4. Edit the **Name** and **Public** visibility as desired.

5. Click the **Update Snapshot** button.

Delete an Instance Snapshot

To delete an instance snapshot

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Snapshots** to display a list of existing snapshots in the project.

3. Select the snapshot to delete and click , **Delete Snapshot**.

To delete multiple instance snapshots

1. Use the left checkbox(es) to select multiple snapshots. To select all, click the top checkbox in the header row.

2. Click the **Delete Snapshots** button.

3. You are prompted to confirm your decision.

Create a Volume Snapshot

Once you have set up a volume, you can create a snapshot to preserve the current state of the volume. Volume Snapshots can then be used to start up another VM in the future.

To create a volume snapshot

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Snapshots**.

3. Click the **Create Volume** button.

4. Enter a **Name** for the snapshot., and optionally a **Description**.

5. Click the **Create Volume** button. The new snapshot is added to the list alphabetically.

Delete a Volume Snapshot

Deleting a volume snapshot cannot be undone.

To delete a volume snapshot

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Snapshots**.

3. Select the Volume Snapshot to delete and click ▼ , **Delete Volume Snapshot**.

To delete multiple volume snapshots

1. Use the left checkbox(es) to select multiple volume snapshots. To select all, click the top checkbox in the header row.

2. Click the **Delete Volume Snapshots** button.

3. You are prompted to confirm your decision.

View Project Access & Security

This tab provides a way to view and manage the following elements in the project:

- **Floating IPs** - Shows IP addresses that have been allocated for this particular project. These available Floating IPs can be dynamically associated and disassociated to a project. Floating IPs that are associated with an instance show the Instance Name and can be disassociated as needed. Association and disassociation are dynamic, as Floating IPs get associated and then released back to the pool which makes them available for use to other projects.

- **Security Groups** - Security groups provide a measure of network isolation between different VMs, as they limit the ports into which network traffic can enter the group.

- **Keypairs** - Keypairs are SSH security keys that provide authentication and allow you to exchange traffic securely within the VM.

To view access and security elements

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Access & Security**.

Associate/Disassociate Floating IPs

Associating a Floating IP assigns the address to a particular VM instance.

To associate a floating IP

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**.

2. Click **Access & Security**.

3. From the Floating IPs list, select an address where the **Associate Floating IP** button is available. If there are no Floating IPs available, see "Allocate an IP to a Project" below

4. Click the **Associate Floating IP** button.

To disassociate a floating IP

Disassociating makes a floating IP available for association within the same project. **Releasing a Floating IP** releases to it the pool and makes it available to all projects.

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**

2. Click **Access & Security**.

3. From the Floating IPs list, select an address where the **Disassociate Floating IP** button is available.

4. Click the **Disassociate Floating IP** button.

Allocate an IP to a Project

Allows you to allocate additional Floating IP addresses to the project. This process does not assign the address to a particular instance in the project. Once allocated, these addresses can be "Associate/Disassociate Floating IPs " on previous pageto a particular instance.

To allocate a floating IP to a project

1. In the left navigation frame of the **Project** tab, select the project from **Current Project.**

2. Click **Access & Security**.

3. From the Floating IPs list, select the **Allocate IP To Project** button.

4. Select from the pool and then click the **Allocate IP** button.

Release Floating IPs

A Floating IP Address associated to a particular project can be released back to the pool and made available to other projects.

To release a floating IP

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**.

2. Click **Access & Security**.

3. From the Floating IPs list, select the address and click ▼, **Release Floating IP**.

Managing Security Groups with Rules

Security Groups are one way to provide a measure of network isolation between different VMs, as they limit the ports into which network traffic can enter the group. Within a security group you can add, edit, and delete rules to specify communications to other ports.

To create a security group

1. In the left navigation frame of the **Project** tab, select the project from **Current Project.**

2. Click **Access & Security**.

3. From the Security Groups list, select the **Create Security Group** button.

4. Enter a **Name** and **Description**.

5. Click the **Create Security Group** button. The group is created and added to the list.

To add security group rules

1. From the Security Groups list, select the group to edit.

2. Click the **Edit Rules** button. The top of the edit screen lists the existing rules; the bottom is where you add.

3. Complete the fields in the **Add Rule** section:
   a. IP Protocol

   b. From Port - identifies the source port (Ex: 80)

   c. To Port - the port where VMs inside the security group would receive (Ex: 80)

   d. Source - identifies the IP addresses from which the group can receive networking traffic

   e. CIDR - the format for specifying a range of IP addresses, which includes how many to create

4. Click the **Update Security Group Rules** button.

To delete security group rules

1. From the Security Groups list, select the group to edit.

2. Click the **Edit Rules** button. The top of the edit screen lists the existing rules.

3. Check the box(es) for the rule(s) you want to delete, then click the **Delete Rule** button.

4. Click the **Update Security Group Rules** button.

To delete a security group

You cannot delete the **default** Security Group.

1. Select the security group to delete, then click [icon], **Delete Security Group**.

2. The system prompts you for confirmation. Click **Delete Security Group**.

To delete multiple security groups

1. Use the left checkbox(es) to select multiple groups. To select all, click the top checkbox in the header row.

2. Click the **Delete Security Groups** button.

3. You are prompted to confirm your decision.

Managing Keypairs

Keypairs are SSH security keys, 512 bits long, that provide authentication and allow you to exchange traffic securely within a VM. These credentials are injected into images when they are launched.

Creating a new keypair registers the public key and downloads the private key (a .pem file), allowing you to log into the VM. The public key gets installed on the VM; the private key is used to log in.

To view project keypairs

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Access & Security**. Existing Keypairs are listed in the last grouping in this section.

To create a keypair

1. Click the **Create Keypair** button.

2. Enter the **Keypair Name**. Keypair names can only contain letters, numbers, underscores, and hyphens.

3. Click the **Create Keypair** button. The credentials are downloaded onto your machine.

To import a keypair

1. Click the **Import Keypair** button.

2. Enter the **Keypair Name**. Keypair names can only contain letters, numbers, underscores, and hyphens.

3. Enter the **Public Key**.

4. Click the **Import Keypai**r button.

To delete a keypair

1. Select the keypair to delete, then click **Delete Keypair**.

2. The system prompts you for confirmation. Click **Delete Keypair**.

To delete multiple keypairs

1. Use the left checkbox(es) to select multiple groups. To select all, click the top checkbox in the header row.

2. Click the **Delete Keypairs** button.

3. You are prompted to confirm your decision.

View Topologies

Topologies represent the virtual infrastructures that have been realized when an Infrastructure Design document is launched in the Cloud Installation Dashboard.

To view topologies

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**.

2. Click **Topologies**.

3. You will see **Active** states when successfully launched doc and no Task and last updated column is timestamp.

4. If unsuccessful, then **State** displays **Failed**. The Task column displays the error for the failed activity, and Last Updated column displays the last time Eve or Eden updated that task.

To view content

1. Click a linked **Topology Name**.

2. The **Overview** tab displays topology details.

3. The **Content** tab displays the design document launched in xml format. **Export** allows you to download the design document to your computer.

4. The **Job** tab displays the activity that occurred when the design document was launched. Click **Refresh** to get an updated view of launch activity.

Launch a Topology

Allows you to launch the latest version of a design document.

To launch a topology

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**.

2. Click **Topologies**.

3. Click the **Launch Topology** button.

4. Select a design **Document** (required).

5. Enter a meaningful **Infrastructure Name** (required).

6. Click the **Launch** button. If successful, a green status box confirms the launch and the topology displays in list. The **State** column will display Build, followed by Active if successful; Failed if unsuccessful.

Terminate a Topology

Terminating a topology stops the VM.

To terminate a topology

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**.

2. Click **Topologies**.

3. Select the Topology to terminate, and click the **Terminate Topology** button

4. The **State** changes to Inactive.

Delete a Topology

Deleting a topology stops the VM and removes it permanently. This action cannot be undone.

To delete a topology

1. In the left navigation frame of the **Project** tab, select the project from **Current Project**.

2. Click **Topologies**.

3. Select the Topology to delete, and click the **Delete Topology** button.

4. The topology no longer displays in the list.

View Documents

A topology document is the result of performing a provision from a design document. Documents viewable in the Cloud Installation Dashboard **Documents** tab are the infrastructure topology templates and resource binding documents (design documents) that are used for provisioning. Documents are stored in Focus and launched through Eve (see).

An Administrator can view, promote, and set documents to public; a user can view only **Public** documents.

To view documents

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents** to view the loaded documents:

Field	Description
Document Name	The name of the document.
Version	Shows the promoted version number. Only an Administrator can promote a document version. Multiple versions of a document can exist, but only one is promoted (either the most recent or specifically set to another version).

Field	Description
Document Type	There are four different types of documents that can be loaded: • **Infrastructure Topology** - describe the set of components that are included in a particular infrastructure topology and how those components are connected with each other. Infrastructure topology documents are created in Topology Designer. • **Infrastructure Types** - describes the set of component types that can be used in a topology documents interacting with Eve. A fixed document that is installed only once. • **Infrastructure Design** - the resource binding documents with actual property values defined. In a complex binding document, you would see one binding element for every node template that exists in the infrastructure topology template • **Infrastructure Metadata** - these documents show the XY pixel placements of components generated in Topology Designer.
Public	If **Public** is **True**, the promoted version is available to all users.
Project	The project into which the document is loaded.
Actions	Available actions depend on the document type.

View Document Details

Document details is a read-only view of the following for all document types:

- General Information
- Version(s)
- Properties

To view document details

1. In the left navigation frame of the **Projects** tab, select the project to view from **Current Project**.

2. Click **Documents**.

3. Click the link on the **Document Name.** The **Document Detail Overview** displays.

4. See "Edit Document Details " on next page to update document properties; see "Edit an Infrastructure Design" on page 73for information on updating Binding Details.

View Document Content

Used to view the URL, the promoted version number, and the xml content of a selected document. Content describes one node type.

To view document content

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents**.

3. Click the linked **Document Name**.

4. Click the **Content** tab to display the xml content.

Edit Document Details

You can edit the following document details:

- Name

- Public visibility

- Properties - Describe metadata about the document.

You cannot change the document **Type**.

Properties describe metadata about the document.

Edit details using the following steps or by "View Document Details" on previous page.

To edit document details

1. In the left navigation frame of the Project tab, click **Documents**.

2. Select the document to edit and click the **Edit Details** button.

3. Update the **Name** and **Public** if desired. Public indicates whether the document is visible to users outside the project. The **Type** field cannot be edited.

4. In the **Property Name** and **Value** section you have the following options:
   a. Edit existing values

   b. Add properties by clicking the **Add Properties** button. This adds a new entry line.

   c. Delete properties by checking the box and then clicking the **Delete Properties** button

5. Click the **Update Details** button.

Export Document Content

Allows you to export the xml content of a document to an external file. If content is revised outside the Cloud Installation Dashboard the new version must be imported.

To export document content

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents**.

3. Click the linked **Document Name**.

4. Click the **Content** tab to display the xml content.

5. Click the **Export Content** button. Depending on your browser setup, the xml file will open automatically or become available to open.

6. If necessary, edit and save the file.

7. Click the **Import New Version** button to "Import a New Document Version" below.

Import a New Document Version

If xml document content is edited outside the Cloud Installation Dashboard, the new version can be imported.

To import a new document version

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents**.

3. Click the linked **Document Name**.

4. Click the **Content** tab to display the current xml content.

5. Click the **Import New Version** button.

6. Under **Contents**, click **Choose File**. (Name is not editable.)

7. Select an xml file to import and click **Open**. The file name displays in place of the **Choose File** button.

8. Click **Import New Version**. The Documents list redisplays and the version number increments.

9. To view details about when a document was last updated, see "View Document Details" on page 69.

Infrastructure Topology

In Cloud Connector, an infrastructure topology is a logical and visual representation of the layout of networking resources and the connectivity of those resources to each other. When you create a topology template in the Topology Designer, it gets published to Focus so that it can be used for infrastructure designs.

A topology template can be used to create physical or virtual environments. A single topology template can be used to create multiple infrastructure designs as the template becomes associated with a resource pool.

Topology templates are saved as XML documents that conform to OASIS Topology and Orchestration Specification for Cloud (TOSCA) format.

For information about how to create and modify an infrastructure topology, see the HP CDA Topology Designer documentation that is available on the HP Software Product Manuals website at *http://h20230.www2.hp.com/selfsolve/manuals*.

- General-access documentation requires that you register for an HP Passport and sign-in.

- Restricted-access documentation requires that you have an active HP support agreement ID (SAID) and an HP Passport sign-in.

Related Topic: "Infrastructure Design" below.

Infrastructure Design

In Cloud Connector, an infrastructure design is a description of the properties and values that can be bound to those properties when an infrastructure topology (template) is associated with a resource pool. These descriptions are saved in an XML document in an OASIS Topology and Orchestration Specification for Cloud (TOSCA) format.

Infrastructure designs are used to provision an Infrastructure as a Service (IaaS). An infrastructure design is imported into HP Continuous Delivery Automation (HP CDA) to create a platform service that can then be published as a platform service offering.

You create an infrastructure design by associating it with an infrastructure topology (template). Your infrastructure design then gets published (promoted) to Focus so that it can be used as a platform service in HP CDA.

Related Topics: "Create an Infrastructure Design" below and "Infrastructure Topology" above.

Create an Infrastructure Design

Use the Create Infrastructure Design wizard to define a resource binding document, which contains networking resources that align with the infrastructure topology. A resource binding document associates a certain infrastructure topology with a certain cloud provider. In a complex resource binding document, there is one binding element for each node in the infrastructure topology template.

Best Practice: Before you create an infrastructure design, make sure there is a relative infrastructure topology and resource pool in Peer. If a topology does not exist in the repository, you must create one before you can create an infrastructure design that will be associated with it.

> **Note:** During Cloud Administration Dashboard setup and configuration, network resources are organized into regions that are governed by user security rules. To provision an infrastructure design, the design must be associated with a resource pool that you have security access to provision from that region.

To create an infrastructure design:

1. Use the following navigation path to open the Create Infrastructure Design wizard:

   Project > Manage Topology > Documents > Create Infrastructure Design

2. Enter a **Name** for the resource binding document.

3. Select an **Infrastructure Topology** from the drop-down list. These are topologies that were created using the Create Infrastructure Topology view in Topology Designer.

4. Select a **Resource Pool** from the drop-down list. These are resource pools that were created using the Create Resource Pool wizard in the Cloud Administration Dashboard.

5. Click **Next** to go to the next step.

6. *(Optional)* Click **Cancel** to exit the wizard without saving the resource binding document.

7. In the Binding Details view, enter information for the **Server Group**, **Volume Group**, **Network Segment**, **Configuration Manager Service**, and **Relationships**. Values that are marked Modifiable are ones that will override existing values when you change them. These are values that were originally defined when a platform was created from a topology template in HP Continuous Delivery Automation (HP CDA).

8. Click **Create Infrastructure Design** to save your design. Wait for the Success message that confirms your design was saved.

9. *(Optional)* Click **Cancel** to exit the wizard without saving the infrastructure design.

Related Topics: "Create a Resource Pool" on page 76. and "Infrastructure Topology" on previous page.

Edit an Infrastructure Design

Use the Edit Contents view to modify a resource binding document, which contains networking resources that align with the infrastructure topology. A resource binding document associates a certain infrastructure topology with a certain cloud provider. In a complex resource binding document, there is one binding element for each node in the infrastructure topology template.

To edit an infrastructure design:

1. Use the following navigation path to open the Edit Contents view:

   Project > Manage Topology > Documents > Document Type > Edit Content

2. In the Binding Details section, change information about the **Server Group**, **Volume Group**, **Network Segment**, **Configuration Manager Service**, and **Relationships**. Values that are

marked Modifiable are ones that will override existing values when you change them. These are values that were originally defined when a platform was created from a topology template in HP Continuous Delivery Automation (HP CDA).

3. Click **Update Contents** to save your design. Wait for the Success message that confirms your design was saved.

4. *(Optional)* Click **Cancel** to exit the Edit Contents view without saving your changes.

Launch an Infrastructure Design

Launching an Infrastructure Design document provisions that version to the cloud. The design document is sent to Eve, and Eve parses and makes calls into Cloud Connector to create the VM with specifications from the binding document.

To launch a design document

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents.**

3. Select the Infrastructure Design document.

4. Click the **Launch** button. The green status box at the upper right confirms the successful launch.

Import a Document

Allows you to create any of the four document types by importing contents from an external xml file and adding properties.

To import a document

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents.**

3. Click the **Import Document** button.

4. Enter a **Name**.

5. Select a **Document Type**.

6. Click the **Choose File** button to import a topology or design document file.

7. Click **Open** to return to the Import Document window.

8. Check **Public** if you want the document to be visible to users outside the project.

9. Click **Add Properties** if desired (properties are not required).

10. To delete properties, use the checkbox and then click **Delete Properties**.

11. Click the **Import Document** button. If successful, the document displays in the Documents list.

Promote a Document Version

Promoting can only be done by a Domain Administrator. The promoted version of a topology document is presented to the user as the default version, whether it is the most recent version or specifically set to an earlier version. Only versions made Public are available to users. The following infrastructure document types can be promoted:

- Topology

- Design

- Types

For more detailed information about the various versions of a document, see "View Document Details" on page 69.

To promote a document version

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents.**

3. Select the document type to promote and click [▼] , **Promote Version**.

4. Select the **Default Version**; the **Name** field is not editable

5. Click the **Promote Version** button.

6. The documents list redisplays with the new promoted version and a green status box in the upper right indicates the action was successful.

Delete a Document Version

This option allows you to delete the entire document or any non-default specific version of document contents. You cannot delete a promoted, public version of a document.

To delete a document version

1. In the left navigation frame of the **Project** tab, select the **Current Project**.

2. Click **Documents.**

3. Select the document to delete and click ▼, **Delete Version**.

4. Select the version number or All Versions.

5. Click the **Delete Version** button. Without confirmation, the document(s) are deleted and the green status box indicates success.

Resource Pools

Resource pools are used to access various cloud providers. For multiple regions, one resource pool is required for each region.

Peer is the service used for resource pool registry. Every time you want to access a different cloud provider, you must define a new record in the Peer Resource Pool Registry. These records then define how to access a particular cloud provider.

Related Topics: "Create a Resource Pool" below, "Edit a Resource Pool" on next page, "Domains" on page 22 and "Regions" on page 35.

Create a Resource Pool

Use the Create Resource Pool wizard to specify resource pool definitions that will be used during topology provisioning.

To create a resource pool:

1. Use the following navigation path to open the Create Resource Pool wizard:

    Project > Manage Topology > Resource Pools > Create Resource Pool

2. From the drop-down list, select a **Cloud Type** to create a resource pool when you are using local (on premise services) or HP Cloud services. When you specify a **Cloud Type**, information in the remaining sections is automatically populated. If you do not select a **Cloud Type**, you must enter information in the remaining sections.

-   **HP Cloud Infrastructure - OpenStack (Essex)**: Provides values for the Cloud Connector environment. Provider Credentials are not required.

-   **HP Cloud - OpenStack (Diablo)**: Provides values for the HP Cloud Services environment. Provider Credentials are required.

-   **Custom - OpenStack (Essex)**: Provides maximum flexibility when you are creating a resource pool. Details and Provider Credentials are enabled.

1. From the drop-down list, select a **Compute Region** to create a resource pool when you are using local (on premise services) or HP Cloud services. If you do not select a **Compute Region**, you must enter information in the remaining fields.

2. In the resource pool **Details** section, enter a **Name**. The **Type**, **Version**, **Region ID**, and **Domain URL** are derived from the **Cloud Type** and **Compute Region** you selected.

3. In the **Provider Credentials** section, enter a **Tenant ID**, **User Name**, and **Password**. Enter the password again in **Confirm Password** to validate.

- If you are using a local (on premise service), such as OpenStack, you do not need an account on that service to create and edit resource pools.

- If you are using a remote service, such as HP Cloud or Rackspace, you must have an account on that cloud provider's system to create and edit resource pools.

4. Click **Create Resource Pool** to save it. Wait for the Success message that confirms your changes were saved.

5. *(Optional)* Click **Cancel** to exit the Create Resource Pool wizard without saving.

Related Topics: "Edit a Resource Pool" below and "Edit a Resource Pool Service" on next page.

Edit a Resource Pool

Use the Edit Resource Pool view to update resource pool definitions that will be used during topology provisioning. In common use cases, you can:

- Change the name of the resource pool.

- Switch to a new region.

- Update the OpenStack server domain URL.

- Modify different provider credentials when the resource is an HP Cloud.

- Modify values that were not accurate when you originally entered them.

> **Note:** If you are using HP Cloud and your password on that system changed, you must also change it the Edit Resource Pool view.

To update resource pool definitions:

1. Use the following navigation path to open the Edit Resource Pool view:

   Project > Manage Topology > Resource Pools > Edit Resource Pool

2. In the **Details** section, modify the **Name**, **Type**, **Version**, **Region ID**, and **Domain URL**.

3. Depending on the Cloud Type, in the **Provider Credentials** section, modify the **Tenant ID**, **User Name**, and **Password**. Enter the password again in **Confirm Password** to validate. See "Create a Resource Pool" on previous page.

4. Click **Update Resource Pool** to save your changes. Wait for the Success message that confirms your changes were saved.

5. *(Optional)* Click **Cancel** to exit the Edit Resource Pool view without saving your changes.

Related Topic: "Edit a Resource Pool Service" on next page.

Delete a Resource Pool

> **Caution:** Make sure you really want to delete a resource pool. Deleted resource pools cannot be restored. You have to recreate them.

Use the Resource Pools view to delete a single resource pool or multiple resource pools from Peer.

Best Practice: Delete the resource pool *after* deleting all services that were using that resource pool. If the environment that the resource pool no longer exists, delete the resource pool.

> **Note:** In Cloud Administration Dashboard, you can delete a resource pool even if a service is provisioned using that resource pool. Also, after the resource pool is deleted, the de-provisioning operation will still succeed.

To delete a single resource pool:

1. In the Resource Pools view, highlight one Resource Pool Name.

2. In the Actions column, in the Edit Resource Pool drop-down list, select the **Delete Resource Pool** action.

To delete multiple resource pools:

1. In the Resource pools view, select the check box next to multiple Resource Pool Names.

2. Click the **Delete Resource Pools** button.

Edit a Resource Pool Service

Use the Edit Resource Pool Services view to add and delete services to and from Peer.

A service definition is a certain model/version of an image/flavor combination. If you find problems in particular model/version, you might be required to add a new resource pool service.

Best Practice: To prevent anyone from using the previous resource pool service that caused problems, be sure and delete it from Peer.

To edit a resource pool service:

1. Use the following navigation path to open the Edit Resource Pool Services view:

    Project > Manage Topology > Resource Pools

2. In the Resource Pools view, from the Edit Resource Pool drop-down list, select the **Edit Services** action.

3. *(Optional)* In the Edit Resource Pool Services view, click **Add Service** and then enter the **Name**, **Type**, **Model**, **Version**, **Flavor ID**, and **Image ID**. Click **Update Resource Pool Services** to save your changes.

4. *(Optional)* In the Edit Resource Pool Services view, select the check boxes next to the Names of the services you want to remove and then click **Delete Services**. Click **Update Resource Pool Services** to save your changes.

5. Wait for the Success message that confirms your changes were saved.

6. *(Optional)* Click **Cancel** to exit the Edit Resource Pools Services view without saving your changes.