

Meghdoot-Openstack

User manual Version-0.01

## Table of Contents

Meghdoot -Quickstart (Single node installation) How to.....	3
Meghdoot -Quickstart (Multi-node installation) How to.....	5
About the Dashboard.....	11
OpenStack dashboard —Project tab.....	12
Compute tab.....	13
Network tab.....	13
OpenStack dashboard — Admin tab.....	14
System tab.....	14
OpenStack dashboard — Identity tab.....	16
OpenStack dashboard — Settings tab.....	17
Upload and manage images.....	17
Upload an image.....	18
Update an image.....	19
Delete an image.....	20
Configure access and security for instances.....	20
Add a rule to the default security group.....	21
Add a key pair.....	22
Import a key pair.....	22
Allocate a floating IP address to an instance.....	23
Launch and manage instances.....	24
Launch an instance.....	24
Connect to your instance by using SSH.....	29
Track usage for instances.....	30
Create an instance snapshot.....	30
Manage an instance.....	31
Create and manage networks.....	31
Create a network.....	31
Create a router.....	32
Create a port.....	33
Create and manage volumes.....	34
Create a volume.....	34
Attach a volume to an instance.....	35
Detach a volume from an instance.....	36
Create a snapshot from a volume.....	36
Edit a volume.....	36
Delete a volume.....	37
Miscellaneous.....	37
Add Compute Nodes to the existing Single node Setup.....	37
Meghdoot Openstack Cloud utility.....	39

## ***Meghdoot -Quickstart (Single node installation) How to***

Before starting the quick-start cloud setup, make sure that the IP address is assigned properly. Make entry in /etc/network/interfaces (as static IP address).

Edit /etc/network/interfaces file

```
vi /etc/network/interfaces
```

Add the following lines

```
auto <interface name... Eg. eth0>
iface <interface name > inet static
address <IP address>
netmask 255.255.255.0
gateway <gateway IP>
```

Save and close the file.

Run

```
/etc/init.d/networking restart
```

Step 1 : Double click meghdoot-quickstart icon available in the desktop.

Step 2 : You will be prompted to provide the login password. Give the password and press 'Enter'.

Step 3 : A warning dialog box opens. **Click Yes to continue.**

Step 4 : The user is asked to choose between multi node installation/single node installation. **Choose Single node installation.**

Step 5 : The user is asked to choose between Physical machine installation(Cloud installed on physical machine)/Virtual machine installation(Cloud installed on VM.).

Step 6 : A dialog box opens asking whether to initialize database. **Click Yes**

Step 7 : Five of the openstack components(keystone, glance, nova, neutron,cinder) will be registered as service. Each component requires password and email for service registration. A dialog box opens and asks 'Would you like to set same password and email for all components ? '

Step 8 : **If you answered yes**, then a form is displayed to enter some values.

**If you are not sure what are the values that need to be given for each field, Enter the following default values.**

Keystone token : admin

Admin tenant name : admin

Service tenant name : service

Horizon username : admin

Horizon password : admin

Admin password : admin - this value will be set as password for all the component's admin user

Admin Email Id : <mail id> - this value will be set as mail id for all the component's admin user

Neutron n/w name : phynet1

VLAN start : 20

VLAN end : 30

Check the values and **Click OK**.

**If you answered no**, then a form is displayed to enter some values. Here you will find separate password and email text entry fields for each component

**If you are not sure what are values that need to be given for each field, Enter the following default values.**

Keystone token : admin

Admin tenant name : admin

Service tenant name : service

Horizon username : admin

Horizon password : admin

Keystone admin mail id : <mail id>

Glance admin password : glance

Glance admin mail id : <mail id>

Nova admin password : nova

Nova admin mail id : <mail id>

Neutron admin password : neutron

Neutron admin mail id : <mail id>

Neutron n/w name : phynet1

VLAN start : 20

VLAN end : 30

Cinder Admin password : cinder

Cinder admin mail id : <mail id>

Check the values and **Click OK**.

All the components will be registered one by one. Wait for the process to finish.

Step 9 : A information dialog is displayed. **Click OK**.

Step 10 : Firefox opens and the openstack dashboard login screen is displayed. Login using the horizon credentials.(default admin & admin)

Note : Do not press Ctrl + C in the terminal where the script is executed.

## ***Meghdoot -Quickstart (Multi-node installation) How to***

Before starting the quick-start cloud setup, make sure that the IP address is assigned properly. Make entry in /etc/network/interfaces (as static IP address).

Edit /etc/network/interfaces file

```
vi /etc/network/interfaces
```

Add the following lines

```
auto <interface name... Eg. eth0>  
iface <interface name > inet static  
address <IP address>  
netmask 255.255.255.0  
gateway <gateway IP>
```

Save and close the file.

Run

```
/etc/init.d/networking restart
```

Step 1 : Double click meghdoot-quickstart icon available in the desktop.

Step 2 : You will be prompted to provide the login password. Give the password and press “Enter”

Step 3 : A warning dialog box opens. **Click Yes to continue.**

Step 4 : The user is asked to choose between multi node installation/single node installation. **Choose Multi node installation.**

Step 5 : The user is asked to choose between Physical machine installation(Cloud installed on physical machine)/Virtual machine installation(Cloud installed on VM.).

Step 6 : A dialog box opens and prompts for the Compute node IP.

Step 7 : A dialog box opens and asks for compute node's username and password. Give the username and password and Click OK.

Step 8 : Background script starts to run in compute node. Do not provide input when prompted for password during the process. A information box is displayed “SSH login test successful”

Step 9 : A dialog box opens and asks to choose compute components. The options are neutron-agents and cinder-volume components. You can either leave the options or choose one or choose both.

Step 10 : A dialog box opens asking whether to initialize database. **Click Yes**

Step 11 : Five of the openstack components(keystone, glance, nova, neutron,cinder) will be registered as service. Each component requires password and email for service registration. A dialog box opens and asks 'Would you like to set same password and email for all components ? '

Step 12 : **If you answered yes**, then a form is displayed to enter some values.

**If you are not sure what are the values that need to be given for each field, Enter the following default values.**

Keystone token : admin

Admin tenant name : admin

Service tenant name : service

Horizon username : admin

Horizon password : admin

Admin password : admin - this value will be set as password for all the component's admin user

Admin Email Id : <mail id> - this value will be set as mail id for all the component's admin user

Neutron n/w name : phynet1

VLAN start : 20

VLAN end : 30

Check the values and **Click OK**.

**If you answered no**, then a form is displayed to enter some values. Here you will find separate password and email text entry fields for each component

**If you are not sure what are values that need to be given for each field, Enter the following default values.**

Keystone token : admin

Admin tenant name : admin

Service tenant name : service

Horizon username : admin

Horizon password : admin

Keystone admin mail id : <mail id>

Glance admin password : glance

Glance admin mail id : <mail id>

Nova admin password : nova

Nova admin mail id : <mail id>

Neutron admin password : neutron

Neutron admin mail id : <mail id>

Neutron n/w name : phynet1

VLAN start : 20

VLAN end : 30

Cinder Admin password : cinder

Cinder admin mail id : <mail id>

Check the values and **Click OK**.

All the components will be registered one by one. During this process, some of the scripts will be run on the compute node. Do not provide input when prompted for password during the process and Wait for the process to finish.

Step 13 : A information dialog is displayed. **Click OK**.

Step 14 : Firefox opens and the openstack dashboard login screen is displayed. Login using the horizon credentials.(default admin & admin)

Note : Do not press Ctrl + C in the terminal where the script is executed.



## ***About the Dashboard***

The dashboard is generally installed on the controller node.

1. Ask the cloud operator for the host name or public IP address from which you can access the dashboard, and for your user name and password. If the cloud supports multi-domain model, you also need to ask for your domain name.
2. Open a web browser that has JavaScript and cookies enabled.
3. In the address bar, enter the host name or IP address for the dashboard, for example `https://ipAddressOrHostName/`.
4. On the Log In page, enter your user name and password, and click Sign In. If the cloud supports multi-domain model, you also need to enter your domain name.

The top of the window displays your user name. You can also access the Settings tab (OpenStack dashboard — Settings tab) or sign out of the dashboard.

The visible tabs and functions in the dashboard depend on the access permissions, or roles, of the user you are logged in as.

- If you are logged in as an end user, the Project tab (OpenStack dashboard — Project tab) and Identity tab (OpenStack dashboard — Identity tab) are displayed.
- If you are logged in as an administrator, the Project tab (OpenStack dashboard — Project tab) and Admin tab (OpenStack dashboard — Admin tab) and Identity tab (OpenStack dashboard — Identity tab) are displayed.

# OpenStack dashboard —Project tab

Projects are organizational units in the cloud, and are also known as tenants or accounts. Each user is a member of one or more projects. Within a project, a user creates and manages instances.

From the Project tab, you can view and manage the resources in a selected project, including instances and images. You can select the project from the drop down menu at the top left. If the cloud supports multi-domain model, you can also select the domain from this menu.

The screenshot shows the OpenStack dashboard interface. At the top left is the OpenStack logo and a user menu for 'admin'. A sidebar on the left contains navigation links for Project, Compute, Overview (highlighted), Instances, Volumes, Images, Access & Security, Network, Orchestration, Object Store, Admin, and Identity. The main content area is titled 'Overview' and 'Limit Summary'. It features seven circular gauges representing resource usage: Instances (Used 0 of 10), VCPUs (Used 0 of 20), RAM (Used 0 of 51,200), Floating IPs (Used 0 of 50), Security Groups (Used 1 of 10), Volumes (Used 0 of 10), and Volume Storage (Used 0 of 1,000). Below this is the 'Usage Summary' section, which includes a date range selector (From: 2015-11-01, To: 2015-11-25) and a 'Submit' button. The usage summary text reads: 'Active Instances: 0 Active RAM: 0Bytes This Period's VCPU-Hours: 0.00 This Period's GB-Hours: 0.00 This Period's RAM-Hours: 0.00'. A 'Download CSV Summary' button is located to the right. At the bottom, there is a table with columns for Instance Name, VCPUs, Disk, RAM, and Time since created. The table currently shows 'No items to display.' and 'Displaying 0 items'.

Instance Name	VCPUs	Disk	RAM	Time since created
No items to display.				
Displaying 0 items				

From the Project tab, you can access the following categories:

### ***Compute tab***

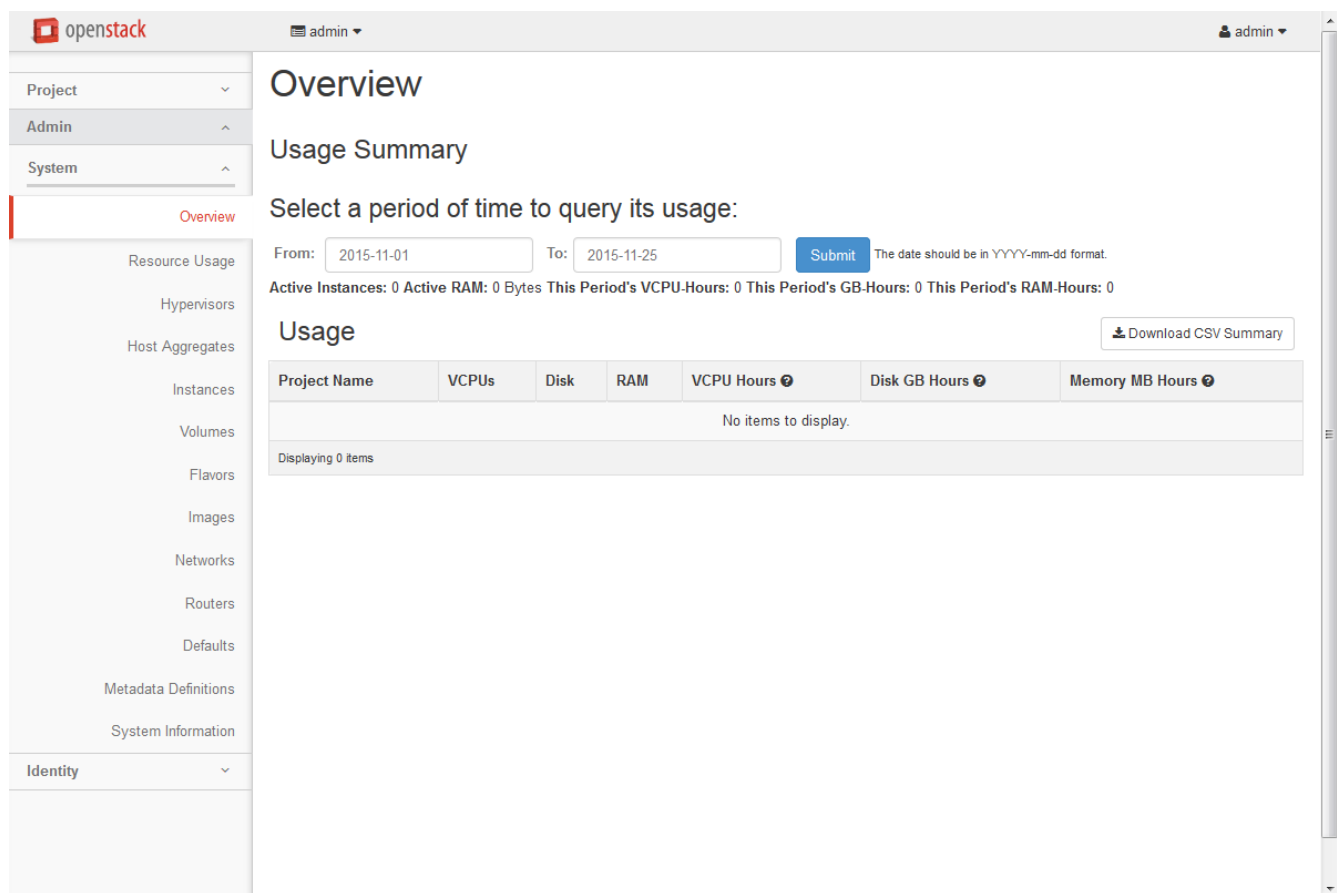
- Overview: View reports for the project.
- Instances: View, launch, create a snapshot from, stop, pause, or reboot instances, or connect to them through VNC.
- Volumes: Use the following tabs to complete these tasks:
  - Volumes: View, create, edit, and delete volumes.
  - Volume Snapshots: View, create, edit, and delete volume snapshots.
- Images: View images and instance snapshots created by project users, plus any images that are publicly available. Create, edit, and delete images, and launch instances from images and snapshots.
- Access & Security: Use the following tabs to complete these tasks:
  - Security Groups: View, create, edit, and delete security groups and security group rules.
  - Key Pairs: View, create, edit, import, and delete key pairs.
  - Floating IPs: Allocate an IP address to or release it from a project.
  - API Access: View API endpoints.

### ***Network tab***

- Network Topology: View the network topology.
- Networks: Create and manage public and private networks.
- Routers: Create and manage routers.

## OpenStack dashboard — Admin tab

Administrative users can use the Admin tab to view usage and to manage instances, volumes, flavors, images, networks and so on.



The screenshot shows the OpenStack Admin dashboard. The top navigation bar includes the OpenStack logo, the user name 'admin', and a dropdown menu. The left sidebar contains a navigation menu with categories like Project, Admin, and System, and sub-items such as Overview, Resource Usage, Hypervisors, Host Aggregates, Instances, Volumes, Flavors, Images, Networks, Routers, Defaults, Metadata Definitions, System Information, and Identity.

The main content area is titled 'Overview' and 'Usage Summary'. It prompts the user to 'Select a period of time to query its usage:' with input fields for 'From: 2015-11-01' and 'To: 2015-11-25', and a 'Submit' button. Below this, it displays summary statistics: 'Active Instances: 0 Active RAM: 0 Bytes This Period's VCPU-Hours: 0 This Period's GB-Hours: 0 This Period's RAM-Hours: 0'. A 'Download CSV Summary' button is also present.

The 'Usage' section features a table with the following columns: Project Name, VCPUs, Disk, RAM, VCPU Hours, Disk GB Hours, and Memory MB Hours. The table currently shows 'No items to display.' and 'Displaying 0 items'.

From the Admin tab, you can access the following category to complete these tasks:

## *System tab*

- Overview: View basic reports.
- Resource Usage: Use the following tabs to view the following usages:
  - Usage Report: View the usage report.
  - Stats: View the statistics of all resources.
- Hypervisors: View the hypervisor summary.
- Host Aggregates: View, create, and edit host aggregates. View the list of availability zones.
- Instances: View, pause, resume, suspend, migrate, soft or hard reboot, and delete running instances that belong to users of some, but not all, projects. Also, view the log for an instance or access an instance through VNC.
- Volumes: Use the following tabs to complete these tasks:
  - Volumes: View, create, manage, and delete volumes.
  - Volume Types: View, create, manage, and delete volume types.
  - Volume Snapshots: View, manage, and delete volume snapshots.
- Flavors: View, create, edit, view extra specifications for, and delete flavors. A flavor is size of an instance.
- Images: View, create, edit properties for, and delete custom images.
- Networks: View, create, edit properties for, and delete networks.
- Routers: View, create, edit properties for, and delete routers.
- Defaults: View default quota values. Quotas are hard-coded in OpenStack Compute and define the maximum allowable size and number of resources.
- Metadata Definitions: Import namespace and view the metadata information.
- System Information: Use the following tabs to view the service information:
  - Services: View a list of the services.
  - Compute Services: View a list of all Compute services.
  - Block Storage Services: View a list of all Block Storage services.
  - Network Agents: View the network agents.
  - Orchestration Services: View a list of all Orchestration services.

# OpenStack dashboard — Identity tab

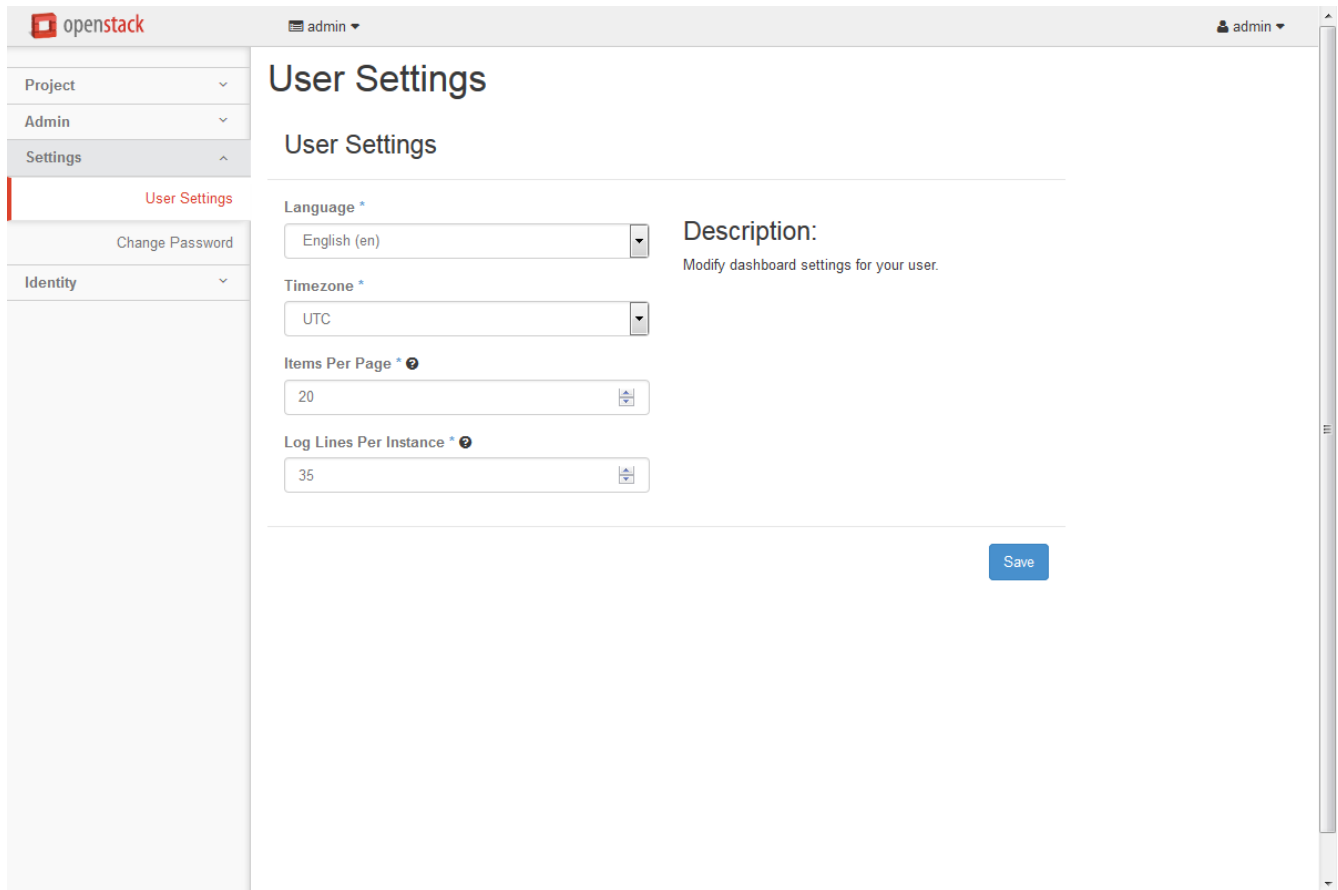
The screenshot shows the OpenStack dashboard interface. At the top left is the OpenStack logo. The user is logged in as 'admin'. The left sidebar contains navigation options: 'Project', 'Admin', and 'Identity'. The 'Identity' section is expanded, showing 'Projects' (highlighted in red) and 'Users'. The main content area is titled 'Projects' and features a search filter, '+ Create Project' button, and 'x Delete Projects' button. Below these is a table with columns: Name, Description, Project ID, Enabled, and Actions. Two projects are listed: 'admin' and 'services'. Each project has a 'Manage Members' dropdown action.

<input type="checkbox"/>	Name	Description	Project ID	Enabled	Actions
<input type="checkbox"/>	admin	admin tenant	8de4dad72776457bb603ff4c666c6f1d	Yes	Manage Members ▾
<input type="checkbox"/>	services	Tenant for the openstack services	03a4921f11f64eaa16f42b98947ab59	Yes	Manage Members ▾

Displaying 2 items

- Projects: View, create, assign users to, remove users from, and delete projects.
- Users: View, create, enable, disable, and delete users.

## OpenStack dashboard — Settings tab



Click the Settings button from the user drop down menu at the top right of any page, you will see the Settings tab.

- User Settings: View and manage dashboard settings.
- Change Password: Change the password of the user.

### ***Upload and manage images***

A virtual machine image, referred to in this document simply as an image, is a single file that contains a virtual disk that has a bootable operating system installed on it. Images are used to create virtual machine instances within the cloud. For information about creating image files, see the OpenStack Virtual Machine Image Guide.

Depending on your role, you may have permission to upload and manage virtual machine images. Operators might restrict the upload and management of images to cloud administrators or operators only. If you have the appropriate privileges, you can use the dashboard to upload and manage images in the admin project.

### ***Upload an image***

Follow this procedure to upload an image to a project:

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Images category.
4. Click Create Image.

The Create An Image dialog box appears.

## Create An Image ✕

**Name** \*

**Description**

**Image Source**

Image Location ▾

**Image Location** ⓘ

**Format** \*

Select format ▾

**Architecture**

**Minimum Disk (GB)** ⓘ

**Minimum RAM (MB)** ⓘ

**Image Location**

**Public**

**Protected**

**Description:**

Currently only images available via an HTTP/HTTPS URL are supported. The image location must be accessible to the Image Service.

**Please note:** The Image Location field **MUST** be a valid and direct URL to the image binary. URLs that redirect or serve error pages will result in unusable images.



5. Enter the following values:

<i>Name</i>	<i>Enter a name for the image.</i>
<i>Description</i>	<i>Enter a brief description of the image.</i>
<i>Image Source</i>	<i>Choose the image source from the dropdown list. Your choices are Image Location and Image File.</i>
<i>Image File or Location</i>	<i>Based on your selection for Image Source, you either enter the location URL of the image in the Image Location field, or browse for the image file on your file system and add it.</i>
<i>Format</i>	<i>Select the image format (for example, QCOW2) for the image.</i>
<i>Architecture</i>	<i>Specify the architecture. For example, i386 for a 32-bit architecture or x86_64 for a 64-bit architecture.</i>
<i>Minimum Disk (GB)</i>	<i>Leave this field empty.</i>
<i>Minimum RAM (MB)</i>	<i>Leave this field empty.</i>
<i>Copy Data</i>	<i>Specify this option to copy image data to the Image service.</i>
<i>Public</i>	<i>Select this check box to make the image public to all users with access to the current project.</i>
<i>Protected</i>	<i>Select this check box to ensure that only users with permissions can delete the image.</i>

6. Click Create Image.

The image is queued to be uploaded. It might take some time before the status changes from Queued to Active.

### ***Update an image***

Follow this procedure to update an existing image.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. Select the image that you want to edit.
4. In the Actions column, click the menu button and then select Edit Image from the list.

5. In the Update Image dialog box, you can perform various actions. For example:
  - Change the name of the image.
  - Select the Public check box to make the image public.
  - Clear the Public check box to make the image private.
6. Click Update Image.

### ***Delete an image***

Deletion of images is permanent and cannot be reversed. Only users with the appropriate permissions can delete images.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Images category.
4. Select the images that you want to delete.
5. Click Delete Images.
6. In the Confirm Delete Images dialog box, click Delete Images to confirm the deletion.

## ***Configure access and security for instances***

Before you launch an instance, you should add security group rules to enable users to ping and use SSH to connect to the instance. Security groups are sets of IP filter rules that define networking access and are applied to all instances within a project. To do so, you either add rules to the default security group or add a new security group with rules.

Key pairs are SSH credentials that are injected into an instance when it is launched. To use key pair injection, the image that the instance is based on must contain the cloud-init package. Each project should have at least one key pair.

If you have generated a key pair with an external tool, you can import it into OpenStack. The key pair can be used for multiple instances that belong to a project.

When an instance is created in OpenStack, it is automatically assigned a fixed IP address in the network to which the instance is assigned. This IP address is permanently associated with the instance until the instance is terminated. However, in addition to the fixed IP address, a floating IP address can also be

attached to an instance. Unlike fixed IP addresses, floating IP addresses are able to have their associations modified at any time, regardless of the state of the instances involved.

### ***Add a rule to the default security group***

This procedure enables SSH and ICMP (ping) access to instances. The rules apply to all instances within a given project, and should be set for every project unless there is a reason to prohibit SSH or ICMP access to the instances.

This procedure can be adjusted as necessary to add additional security group rules to a project, if your cloud requires them.

#### Note

When adding a rule, you must specify the protocol used with the destination port or source port.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Access & Security category. The Security Groups tab shows the security groups that are available for this project.
4. Select the default security group and click Manage Rules.
5. To allow SSH access, click Add Rule.
6. In the Add Rule dialog box, enter the following values:
  - Rule: SSH
  - Remote: CIDR
  - CIDR: 0.0.0.0/0

#### Note

To accept requests from a particular range of IP addresses, specify the IP address block in the CIDR box.

7. Click Add.

Instances will now have SSH port 22 open for requests from any IP address.

8. To add an ICMP rule, click Add Rule.
9. In the Add Rule dialog box, enter the following values:
  - Rule: All ICMP
  - Direction: Ingress
  - Remote: CIDR
  - CIDR: 0.0.0.0/0
10. Click Add.

Instances will now accept all incoming ICMP packets.

### ***Add a key pair***

Create at least one key pair for each project.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Access & Security category.
4. Click the Key Pairs tab, which shows the key pairs that are available for this project.
5. Click Create Key Pair.
6. In the Create Key Pair dialog box, enter a name for your key pair, and click Create Key Pair.
7. Respond to the prompt to download the key pair.

### ***Import a key pair***

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Access & Security category.
4. Click the Key Pairs tab, which shows the key pairs that are available for this project.
5. Click Import Key Pair.
6. In the Import Key Pair dialog box, enter the name of your key pair, copy the public key into the Public Key box, and then click Import Key Pair.
7. Save the \*.pem file locally.

8. To change its permissions so that only you can read and write to the file, run the following command:

```
$ chmod 0600 yourPrivateKey.pem
```

Note

If you are using the Dashboard from a Windows computer, use PuTTYgen to load the \*.pem file and convert and save it as \*.ppk.

The Compute database registers the public key of the key pair.

The Dashboard lists the key pair on the Access & Security tab.

### ***Allocate a floating IP address to an instance***

When an instance is created in OpenStack, it is automatically assigned a fixed IP address in the network to which the instance is assigned. This IP address is permanently associated with the instance until the instance is terminated.

However, in addition to the fixed IP address, a floating IP address can also be attached to an instance. Unlike fixed IP addresses, floating IP addresses can have their associations modified at any time, regardless of the state of the instances involved. This procedure details the reservation of a floating IP address from an existing pool of addresses and the association of that address with a specific instance.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Access & Security category.
4. Click the Floating Ips tab, which shows the floating IP addresses allocated to instances.
5. Click Allocate IP To Project.
6. Choose the pool from which to pick the IP address.
7. Click Allocate IP.
8. In the Floating Ips list, click Associate.
9. In the Manage Floating IP Associations dialog box, choose the following options:

- The IP Address field is filled automatically, but you can add a new IP address by clicking the + button.
- In the Port to be associated field, select a port from the list.

The list shows all the instances with their fixed IP addresses.

10. Click Associate.

Note

To disassociate an IP address from an instance, click the Disassociate button.

To release the floating IP address back into the pool of addresses, click the Release Floating IP option at the Actions column.

## ***Launch and manage instances***

Instances are virtual machines that run inside the cloud. You can launch an instance from the following sources:

- Images uploaded to the Image service.
- Image that you have copied to a persistent volume. The instance launches from the volume, which is provided by the cinder-volume API through iSCSI.
- Instance snapshot that you took.

### ***Launch an instance***

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Instances category.

The dashboard shows the instances with its name, its private and floating IP addresses, size, status, task, power state, and so on.

4. Click Launch Instance.

5. In the Launch Instance dialog box, specify the following values:

**Details tab**

***Availability Zone***

By default, this value is set to the availability zone given by the cloud provider (for example, it could be nova.)

***Instance Name***

Assign a name to the virtual machine.

Note

The name you assign here becomes the initial host name of the server. After the server is built, if you change the server name in the API or change the host name directly, the names are not updated in the dashboard. Server names are not guaranteed to be unique when created so you could have two instances with the same host name.

***Flavor***

Specify the size of the instance to launch.

Note

The flavor is selected based on the size of the image selected for launching an instance. For example, while creating an image, if you have entered the value in the Minimum RAM (MB) field as 2048, then on selecting the image, the default flavor is m1.small.

***Instance Count***

To launch multiple instances, enter a value greater than 1. The default is 1.

***Instance Boot Source***

Your options are:

Boot from image

If you choose this option, a new field for Image Name displays. You can select the image from the list.

### ***Boot from snapshot***

If you choose this option, a new field for Instance Snapshot displays. You can select the snapshot from the list.

### ***Boot from volume***

If you choose this option, a new field for Volume displays. You can select the volume from the list.

### ***Boot from image*** (creates a new volume)

With this option, you can boot from an image and create a volume by entering the Device Size and Device Name for your volume. Click the Delete Volume on Instance Delete option to delete the volume on deleting the instance.

### ***Boot from volume snapshot*** (creates a new volume)

Using this option, you can boot from a volume snapshot and create a new volume by choosing Volume Snapshot from a list and adding a Device Name for your volume. Click the Delete Volume on Instance Delete option to delete the volume on deleting the instance.

### ***Image Name***

This field changes based on your previous selection. If you have chosen to launch an instance using an image, the Image Name field displays. Select the image name from the dropdown list.

### ***Instance Snapshot***

This field changes based on your previous selection. If you have chosen to launch an instance using a snapshot, the Instance Snapshot field displays. Select the snapshot name from the dropdown list.



## ***Volume***

This field changes based on your previous selection. If you have chosen to launch an instance using a volume, the Volume field displays. Select the volume name from the dropdown list. If you want to delete the volume on instance delete, check the Delete Volume on Instance Delete option.

## **Access & Security tab**

### ***Key Pair***

Specify a key pair.

If the image uses a static root password or a static key set (neither is recommended), you do not need to provide a key pair to launch the instance.

### ***Security Groups***

Activate the security groups that you want to assign to the instance.

Security groups are a kind of cloud firewall that define which incoming network traffic is forwarded to instances.

If you have not created any security groups, you can assign only the default security group to the instance.

## **Networking tab**

### ***Selected Networks***

To add a network to the instance, click the + in the Available Networks field.

## **Network Ports tab**

### ***Ports***

Activate the ports that you want to assign to the instance.

### ***Post-Creation tab***

#### Customization Script Source

Specify a customization script that runs after your instance launches.

### **Advanced Options tab**

#### **Disk Partition**

Select the type of disk partition from the dropdown list:

##### ***Automatic***

Entire disk is single partition and automatically resizes.

##### ***Manual***

Faster build times but requires manual partitioning.

#### 6. Click Launch.

The instance starts on a compute node in the cloud.

#### Note

If you did not provide a key pair, security groups, or rules, users can access the instance only from inside the cloud through VNC. Even pinging the instance is not possible without an ICMP rule configured.

You can also launch an instance from the Images or Volumes category when you launch an instance from an image or a volume respectively.

When you launch an instance from an image, OpenStack creates a local copy of the image on the compute node where the instance starts.

When you launch an instance from a volume, note the following steps:

- To select the volume from which to launch, launch an instance from an arbitrary image on the volume. The arbitrary image that you select does not boot. Instead, it is replaced by the image on the volume that you choose in the next steps.

To boot a Xen image from a volume, the image you launch in must be the same type, fully virtualized or paravirtualized, as the one on the volume.

- Select the volume or volume snapshot from which to boot. Enter a device name. Enter vda for KVM images or xvda for Xen images.

## ***Connect to your instance by using SSH***

To use SSH to connect to your instance, use the downloaded keypair file.

Note

The user name is ubuntu for the Ubuntu cloud images on TryStack.

1. Copy the IP address for your instance.
2. Use the ssh command to make a secure connection to the instance. For example:

```
$ ssh -i MyKey.pem ubuntu@10.0.0.2
```

3. At the prompt, type yes.

It is also possible to SSH into an instance without an SSH keypair, if the administrator has enabled root password injection.

### ***Injecting the administrator password***

Compute can generate a random administrator (root) password and inject that password into an instance. If this feature is enabled, users can run ssh to an instance without an ssh keypair. The random password appears in the output of the nova bootcommand. You can also view and set the admin password from the dashboard.

### ***Password injection using the dashboard***

By default, the dashboard will display the admin password and allow the user to modify it.

If you do not want to support password injection, disable the password fields by editing the dashboard's local\_settings.py file.

```
OPENSTACK_HYPERVISOR_FEATURES = {  
...  
    'can_set_password': False,  
}
```

### ***Password injection on libvirt-based hypervisors***

For hypervisors that use the libvirt back end (such as KVM, QEMU, and LXC), admin password injection is disabled by default. To enable it, set this option in `/etc/nova/nova.conf`:

```
[libvirt]  
inject_password=true
```

When enabled, Compute will modify the password of the admin account by editing the `/etc/shadow` file inside the virtual machine instance.

#### Note

Users can only use ssh to access the instance by using the admin password if the virtual machine image is a Linux distribution, and it has been configured to allow users to use ssh as the root user.

### ***Track usage for instances***

You can track usage for instances for each project. You can track costs per month by showing meters like number of vCPUs, disks, RAM, and uptime for all your instances.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Overview category.
4. To query the instance usage for a month, select a month and click Submit.
5. To download a summary, click Download CSV Summary.

### ***Create an instance snapshot***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Compute tab and click the Instances category.
3. Select the instance from which to create a snapshot.

4. In the Actions column, click Create Snapshot.
5. In the Create Snapshot dialog box, enter a name for the snapshot, and click Create Snapshot.

The Images category shows the instance snapshot.

To launch an instance from the snapshot, select the snapshot and click Launch. Proceed with launching an instance.

### ***Manage an instance***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Compute tab and click Instances category.
3. Select an instance.
4. In the menu list in the Actions column, select the state.

You can resize or rebuild an instance. You can also choose to view the instance console log, edit instance or the security groups. Depending on the current state of the instance, you can pause, resume, suspend, soft or hard reboot, or terminate it.

## ***Create and manage networks***

The OpenStack Networking service provides a scalable system for managing the network connectivity within an OpenStack cloud deployment. It can easily and quickly react to changing network needs (for example, creating and assigning new IP addresses).

Networking in OpenStack is complex. This section provides the basic instructions for creating a network and a router.

### ***Create a network***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Network tab and click Networks category.

3. Click Create Network.
4. In the Create Network dialog box, specify the following values.

Network tab

Network Name: Specify a name to identify the network.

Admin State: The state to start the network in.

Create Subnet: Select this check box to create a subnet

You do not have to specify a subnet when you create a network, but if you do not specify a subnet, any attached instance receives an Error status.

Subnet tab

Subnet Name: Specify a name for the subnet.

Network Address: Specify the IP address for the subnet.

IP Version: Select IPv4 or IPv6.

Gateway IP: Specify an IP address for a specific gateway. This parameter is optional.

Disable Gateway: Select this check box to disable a gateway IP address.

Subnet Details tab

Enable DHCP: Select this check box to enable DHCP.

Allocation Pools: Specify IP address pools.

DNS Name Servers: Specify a name for the DNS server.

Host Routes: Specify the IP address of host routes.

5. Click Create.

The dashboard shows the network on the Networks tab.

### ***Create a router***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Network tab and click Routers category.

3. Click Create Router.
4. In the Create Router dialog box, specify a name for the router and External Network, and click Create Router.

The new router is now displayed in the Routers tab.

5. To connect a private network to the newly created router, perform the following steps:
  1. On the Routers tab, click the name of the router.
  2. On the Router Details page, click the Interfaces tab, then click Add Interface.
  3. In the Add Interface dialog box, select a Subnet.

Optionally, in the Add Interface dialog box, set an IP Address for the router interface for the selected subnet.

If you choose not to set the IP Address value, then by default OpenStack Networking uses the first host IP address in the subnet.

The Router Name and Router ID fields are automatically updated.

6. Click Add Interface.

You have successfully created the router. You can view the new topology from the Network Topology tab.

### ***Create a port***

1. Log in to the dashboard.
2. Select the appropriate project from the drop-down menu at the top left.
3. On the Admin tab, click Networks category.
4. Click on the Network Name of the network in which the port has to be created.
5. In the Create Port dialog box, specify the following values.

Name: Specify name to identify the port.

Device ID: Device ID attached to the port.

Device Owner: Device owner attached to the port.

Binding Host: The ID of the host where the port is allocated.

Binding VNIC Type: Select the VNIC type that is bound to the neutron port.

6. Click Create Port.

The new port is now displayed in the Ports list.

## *Create and manage volumes*

Volumes are block storage devices that you attach to instances to enable persistent storage. You can attach a volume to a running instance or detach a volume and attach it to another instance at any time. You can also create a snapshot from or delete a volume. Only administrative users can create volume types.

### **Create a volume**

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Compute tab and click Volumes category.
3. Click Create Volume.

In the dialog box that opens, enter or select the following values.

Volume Name: Specify a name for the volume.

Description: Optionally, provide a brief description for the volume.

Volume Source: Select one of the following options:

- No source, empty volume: Creates an empty volume. An empty volume does not contain a file system or a partition table.
- Image: If you choose this option, a new field for Use image as a source displays. You can select the image from the list.
- Volume: If you choose this option, a new field for Use volume as a source displays. You can select the volume from the list. Options to use a snapshot or a volume as the source for a volume are displayed only if there are existing snapshots or volumes.

Type: Leave this field blank.



Size (GB): The size of the volume in gibibytes (GiB).

Availability Zone: Select the Availability Zone from the list. By default, this value is set to the availability zone given by the cloud provider (for example, us-west or apac-south). For some cases, it could be nova.

4. Click Create Volume.

The dashboard shows the volume on the Volumes tab.

### ***Attach a volume to an instance***

After you create one or more volumes, you can attach them to instances. You can attach a volume to one instance at a time.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Volumes category.
4. Select the volume to add to an instance and click Manage Attachments.
5. In the Manage Volume Attachments dialog box, select an instance.
6. Enter the name of the device from which the volume is accessible by the instance.

#### Note

The actual device name might differ from the volume name because of hypervisor settings.

7. Click Attach Volume.

The dashboard shows the instance to which the volume is now attached and the device name.

You can view the status of a volume in the Volumes tab of the dashboard. The volume is either Available or In-Use.

Now you can log in to the instance and mount, format, and use the disk.

### ***Detach a volume from an instance***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Compute tab and click the Volumes category.
3. Select the volume and click Manage Attachments.
4. Click Detach Volume and confirm your changes.

A message indicates whether the action was successful.

### ***Create a snapshot from a volume***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Compute tab and click Volumes category.
3. Select a volume from which to create a snapshot.
4. In the Actions column, click Create Snapshot.
5. In the dialog box that opens, enter a snapshot name and a brief description.
6. Confirm your changes.

The dashboard shows the new volume snapshot in Volume Snapshots tab.

### ***Edit a volume***

Log in to the dashboard.

1. Select the appropriate project from the drop down menu at the top left.
2. On the Project tab, open the Compute tab and click Volumes category.
3. Select the volume that you want to edit.
4. In the Actions column, click Edit Volume.
5. In the Edit Volume dialog box, update the name and description of the volume.
6. Click Edit Volume.

Note

You can extend a volume by using the Extend Volume option available in the More dropdown list and entering the new value for volume size.

### ***Delete a volume***

When you delete an instance, the data in its attached volumes is not deleted.

1. Log in to the dashboard.
2. Select the appropriate project from the drop down menu at the top left.
3. On the Project tab, open the Compute tab and click Volumes category.
4. Select the check boxes for the volumes that you want to delete.
5. Click Delete Volumes and confirm your choice.

A message indicates whether the action was successful.

## ***Miscellaneous***

### ***Add Compute Nodes to the existing Single node Setup***

Note : It is assumed that the one machine is already installed with Meghdoot-openstack ISO and cloud is configured using the steps mentioned in Meghdoot-quickstart guide. This machine will be the controller node.

Step 1 : Install the Meghdoot-openstack ISO in all the other servers.

Step 2 : **Login to each of the server.** Make sure that the IP address is assigned properly. Make entry in /etc/network/interfaces (as static IP address).

Edit /etc/network/interfaces file

```
vi /etc/network/interfaces
```

Add the following lines

```
auto <interface name... Eg. eth0>
iface <interface name > inet static
address <IP address>
netmask 255.255.255.0
gateway <gateway IP>
```

The sample edited file would look like the following:

```
# The loopback network interface
auto lo
iface lo inet loopback

auto eth0
iface eth0 inet static
address 10.184.48.124
netmask 255.255.255.0
gateway 10.184.48.1
```

Save and close the file.

Run

```
/etc/init.d/networking restart
```

After completing the network configuration on all the servers, you should be able to ping the servers from the controller and vice-versa.

Step 3: Open the terminal in the controller node. Change to root user using `sudo -i`.

Step 4: Type the command `sh /opt/openstack/add_compute.sh`

Step 5: A dialog box opens asking for Compute Node IP which should be added. Give the IP of one of your compute node.

Step 6: A dialog box opens asking for the compute node's username and password. Give the compute node's username and password.

*An automated script runs in the compute node and the compute node will be successfully added.*

### ***Meghdoot Openstack Cloud utility***

[#MeghdootQuickstartSingleNodeInstall](#) Meghdoot openstack cloud utility is a simple GUI where some of the tasks that need to be done from command line are done with minimal user inputs.

It includes

- **Cloud initialization**

Setup a cloud controller in the underlying machine. Single node installation/multi node installation can be done. Refer [Meghdoot-quickstart \(Single node installation\) How to](#) / [Meghdoot-quickstart \(Multi-node installation\) How to](#)

- **Starting/stopping cloud services**

This will stop/start all the currently configured cloud services. For example, If this is a controller node all the services of the controller will be stopped/started.

- **Starting/stopping individual service components**

This will allow a user to start/stop individual services. There might be a situation when a user wants to stop and start a particular service. A separate menu is available for the openstack components. The user can

- Start/Stop Keystone & Glance service components
- Start/Stop Nova service components
- Start/Stop Neutron service components
- Start/Stop Cinder service components
- **Adding a new compute node from the controller machine.**

Refer [Add Compute Nodes to the existing Single node Setup](#)

- **Help**

- **Help Contents**

Information on how to use the Openstack Dashboard along with Meghdoot-openstack quick-start guide

- **About Meghdoot openstack**

The copyright information, the product's current version number and other related details will be available.