RED HAT
SATELLITE
6.4

PROVISIONING GUIDE

A GUIDE TO PROVISIONING PHYSICAL AND VIRTUAL HOSTS ON RED HAT SATELLITE SERVERS.

Red Hat Satellite Documentation Team
satellite-doc-list@redhat.com

Legal Notice

Abstract

The Red Hat Satellite Provisioning Guide has instructions on provisioning physical and virtual hosts. This includes setting up the required network topology, configuring the necessary services, and providing all of the other configuration information to provision hosts on your network.

CHAPTER 1. INTRODUCTION

Provisioning is a process that starts with a bare physical or virtual machine and ends with a fully configured, ready-to-use operating system. Using Red Hat Satellite, you can define and automate fine-grained provisioning for a large number of hosts.

There are many provisioning methods. For example, you can use Satellite Server's integrated Capsule or an external Capsule Server to provision bare metal hosts using both PXE based and non-PXE based methods. You can also provision cloud instances from specific providers through their APIs. These provisioning methods are part of the Red Hat Satellite 6 application life cycle to create, manage, and update hosts.

1.1. PROVISIONING TYPES OVERVIEW

Red Hat Satellite 6 has different methods for provisioning hosts:

Bare Metal Provisioning

Satellite provisions bare metal hosts primarily through PXE boot and MAC address identification. You can create host entries and specify the MAC address of the physical host to provision. You can also boot blank hosts to use Satellite's discovery service, which creates a pool of ready-to-provision hosts. You can also boot and provision hosts through PXE-less methods.

Cloud Providers

Satellite connects to private and public cloud providers to provision instances of hosts from images that are stored with the Cloud environment. This also includes selecting which hardware profile or flavor to use.

Virtualization Infrastructure

Satellite connects to virtualization infrastructure services such as Red Hat Virtualization and VMware to provision virtual machines from virtual image templates or using the same PXE-based boot methods as bare metal providers.

Linux Containers

Satellite can create and manage containers on Red Hat Enterprise Linux Atomic Server.

1.2. PROVISIONING WORKFLOW OVERVIEW

The provisioning process follows a basic workflow:

1. You create a host. Satellite requests an unused IP address from the DHCP Capsule Server that is associated with the subnet. Satellite loads this IP address into the **IP address** field in the Create Host window. When you complete all the options for the new host, submit the new host request.
2. The DHCP Capsule Server that is associated with the subnet reserves an entry for the host.
3. Satellite configures the DNS records:
   - A forward DNS record is created on the Capsule Server that is associated with the domain.
   - A reverse DNS record is created on the DNS Capsule Server that is associated with the subnet.
4. A PXELinux menu is created for the host in the TFTP Capsule Server that is associated with the subnet.
5. The new host requests a DHCP lease from the DHCP server.
6. The DHCP server responds to the lease request and returns TFTP **`next-server`** and **`filename`** options.
7. The host requests the bootloader and menu from the TFTP server.
8. The PXELinux menu and OS installer for the host is returned over TFTP.
9. The installer requests the provisioning template or script from Satellite.
10. Satellite renders the template and returns the resulting kickstart file to the host.
11. The host enters a build process that installs the operating system, registers the host to Satellite, and installs management tools such as **`katello-agent`** and **`puppet`**.
12. The installer notifies Satellite of a successful build in the **`postinstall`** script.
13. The PXELinux menu reverts to a local boot template.
14. The host starts its operating system. If you configured the host to use any Puppet classes, the host configures itself using the modules stored on Satellite.

This workflow differs depending on custom options. For example:

Discovery

If you use the Discovery service, Satellite automatically detects the MAC address of the new host and restarts the host after you submit a request. Note that TCP port 8443 must be reachable by the Capsule to which the host is attached for Satellite to restart the host.

PXE-less Provisioning

After you submit a new host request, you must boot the specific host with a boot disk that you download from Satellite.

Compute Resources

The compute resource creates the virtual machine for the host and returns the MAC address to Satellite. If you use image-based provisioning, the host does not follow the standard PXE boot and operating system installation: the compute resource creates a copy of the image for the new host to use.

Containers

The container provisioning process does not follow the workflow process.

CHAPTER 2. CONFIGURING PROVISIONING CONTEXTS

Before you provision hosts using Satellite, you must understand the placement strategy for hosts in Satellite. This is known as the provisioning context.

2.1. PROVISIONING CONTEXT OVERVIEW

A provisioning context is the combination of an organization and location that you specify for Satellite components. The organization and location that a component belongs to sets the ownership and access for that component.

Organizations divide Red Hat Satellite 6 components into logical groups based on ownership, purpose, content, security level, and other divisions. You can create and manage multiple organizations through Red Hat Satellite 6 and assign components to each individual organization. This ensures the Satellite Server provisions hosts within a certain organization and only uses components that are assigned to that organization. For more information about organizations, see Managing Organizations in the *Content Management Guide*.

Locations function similar to organizations. The difference is that locations are based on physical or geographical setting. Users can nest locations in a hierarchy.

For more information about locations, see Managing Locations in the *Content Management Guide*.

2.2. SETTING THE PROVISIONING CONTEXT

When you set a context, you define which organization and location to use for provisioning hosts.

The organization and location menus are located in the menu bar, on the upper left of the Satellite web UI. If you have not selected an organization and location to use, the menu displays: **Any Organization** and **Any Location**.

Procedure

- To set the provisioning context, click the **Any Organization** and **Any Location** buttons and select the organization and location to use.

Each user can set their default provisioning context in their account settings. Click the user name in the upper right of the Satellite web UI and select **My account** to edit your user account settings.

For CLI Users

While using the CLI, include either **`--organization`** or **`--organization-label`** and **`--location`** or **`--location-id`** as an option. For example:

```
# hammer host list --organization "Default_Organization" --location
"Default_Location"
```

This command outputs hosts allocated for the Default_Organization and Default_Location.

CHAPTER 3. CONFIGURING PROVISIONING RESOURCES

Red Hat Satellite 6 contains provisioning resources that you can use to create a host.

3.1. CREATING OPERATING SYSTEMS

An operating system is a collection of resources that define how Satellite Server installs a base operating system on a host. Operating system entries combine previously defined resources, such as installation media, partition tables, provisioning templates, and others.

Importing operating systems from Red Hat’s CDN creates new entries on the **Hosts** > **Operating Systems** page.

You can also add custom operating systems using the following procedure:

Procedure

1. In the Satellite web UI, navigate to **Hosts** > **Operating systems** and click **New Operating** system.
2. In the **Name** field, enter a name to represent the operating system entry.
3. In the **Major** field, enter the number that corresponds to the major version of the operating system.
4. In the **Minor** field, enter the number that corresponds to the minor version of the operating system.
5. In the **Description** field, enter a description of the operating system.
6. From the **Family** list, select the operating system’s family.
7. From the **Root Password Hash** list, select the encoding method for the root password.
8. From the **Architectures** list, select the architectures that the operating system uses.
9. Click the **Partition table** tab and select the possible partition tables that apply to this operating system.

10. Optional: if you use non-Red Hat content, click the **Installation media** tab and select the installation media that apply to this operating system. For more information, see Section 3.9, “Using Third Party Installation Media”.

11. Click the **Templates** tab and select a **PXELinux template**, a **Provisioning template**, and a **Finish template** for your operating system to use. You can select other templates, for example an **iPXE template**, if you plan to use iPXE for provisioning.

12. Click **Submit** to save your provisioning template.

For CLI Users

Create the operating system using the **`hammer os create`** command:

```
# hammer os create --name "MyOS" \
--description "My_custom_operating_system" \
--major 7 --minor 3 --family "Redhat" --architectures "x86_64" \
--partition-tables "My_Partition" --media "Red_Hat" \
--provisioning-templates "My_Provisioning_Template"
```

3.2. CREATING ARCHITECTURES

An architecture in Satellite represents a logical grouping of hosts and operating systems. Architectures are created by Satellite automatically when hosts check in with Puppet. Basic i386 and x86_64 architectures are already preset in Satellite 6.

Use this procedure to create an architecture in Satellite.

Supported Architectures

Intel x86_64 architecture is supported only for provisioning using PXE, Discovery, and boot disk. For more information, see Red Hat Knowledgebase solution Architectures Supported for Satellite 6 Provisioning.

Procedure

To create an architecture, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Architectures**, and in the Architectures window, click **Create Architecture**.
2. In the **Name** field, enter a name for the architecture.
3. From the **Operating Systems** list, select an operating system. If none are available, you can create and assign them under **Hosts** > **Operating Systems**.
4. Click **Submit**.

For CLI Users

Enter the **`hammer architecture create`** command to create an architecture. Specify its name and operating systems that include this architecture:

```
# hammer architecture create --name "Architecture_Name" \
--operatingsystems "os"
```

3.3. CREATING HARDWARE MODELS

Use this procedure to create a hardware model in Satellite so that you can specify what hardware model a host uses.

Procedure

To create a hardware model, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Hardware Models**, and in the Hardware Models window, click **Create Model**.
2. In the **Name** field, enter a name for the hardware model.
3. Optionally, in the **Hardware Model** and **Vendor Class** fields, you can enter corresponding information for

your system.

4. In the **Info** field, enter a description of the hardware model.

5. Click **Submit** to save your hardware model.

For CLI Users

Create a hardware model using the **`hammer model create`** command. The only required parameter is **`--name`**. Optionally, enter the hardware model with the **`--hardware-model`** option, a vendor class with the **`--vendor-class`** option, and a description with the **`--info`** option:

```
# hammer model create --name "model_name" --info "description" \
--hardware-model "hardware_model" --vendor-class "vendor_class"
```

3.4. USING A SYNCED KICKSTART REPOSITORY FOR A HOST'S OPERATING SYSTEM

Satellite contains a set of synchronized kickstart repositories that you use to install the provisioned host's operating system.

To set up a kickstart repository, complete the following steps:

1. Add the synchronized kickstart repository that you want to use to the existing Content View or create a new Content View and add the kickstart repository.

2. Publish a new version of the Content View where the kickstart repository is added and promote it to a required lifecycle environment. For more information, see Managing Content Views in the *Content Management Guide*.

3. When you create a host, in the **Operating System** tab, for **Media Selection**, select the **Synced Content** check box.

To View the Kickstart Tree

To view the kickstart tree enter the following command:

```
# hammer medium list --organization "your_organization"
```

3.5. CREATING PARTITION TABLES

A partition table is a set of directives that defines the way Satellite Server configures the disks available on a new host. Red Hat Satellite 6 contains a set of default partition tables to use, including a **`Kickstart default`**. You can also edit partition table entries to configure the preferred partitioning scheme, or create a partition table entry and add it to the Red Hat Enterprise Linux operating system entry.

Procedure

To create partition tables, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Partition Tables** and, in the Partition Tables window, click **Create Partition Table**.
2. In the **Name** field, enter a name to represent the partition table.
3. Select the **Default** check box if you want to set the template to automatically associate with new organizations or locations.
4. Select the **Snippet** check box if you want to identify the template as a reusable snippet for other partition tables.
5. From the **Operating System Family** list, select the distribution or family of the partitioning layout. For example, Red Hat Enterprise Linux, CentOS, and Fedora are in the Red Hat family.
6. In the **Template editor** field, enter the layout for the disk partition. For example:

   ```
   zerombr
   clearpart --all --initlabel
   autopart
   ```

You can also use the **Template** file browser to upload a template file.

The format of the layout must match that for the intended operating system. For example, Red Hat Enterprise Linux 7.2 requires a layout that matches a kickstart file.

7. In the **Audit Comment** field, add a summary of changes to the partition layout.
8. Click the **Organizations** and **Locations** tabs to add any other provisioning contexts that you want to associate with the partition table. Satellite adds the partition table to the current provisioning context.
9. Click **Submit** to save your partition table.

For CLI Users

Before you create a partition table with the CLI, create a plain text file that contains the partition layout. This example uses the **`~/my-partition`** file. Create the installation medium using the **`hammer partition-table create`** command:

```
# hammer partition-table create --name "My Partition" --snippet
false \
--os-family Redhat --file ~/my-partition --organizations
"My_Organization" \
--locations "My_Location"
```

3.6. CREATING PROVISIONING TEMPLATES

A provisioning template defines the way Satellite Server installs an operating system on a host.

3.6.1. Types of Provisioning Templates

There are various types of provisioning templates, including:

Provision

The main template for the provisioning process. For example, a kickstart template. For more information about kickstart template syntax, see the Kickstart Syntax Reference in the *Red Hat Enterprise Linux 7 Installation Guide*.

PXELinux, **PXEGrub**, **PXEGrub2**

PXE-based templates that deploy to the template Capsule associated with a subnet to ensure that the host uses the installer with the correct kernel options. For BIOS provisioning, select **PXELinux** template. For UEFI provisioning, select **PXEGrub2**.

Finish

Post-configuration scripts to use when the main provisioning process completes. This is completed as an SSH task. You can use Finishing templates only for imaged-based provisioning in virtual environments. Do not confuse an image with a foreman discovery ISO, which is sometimes called a Foreman discovery image. An image in this context is an install image in a virtualized environment for easy deployment.

Bootdisk

Templates for PXE-less boot methods.

Kernel Execution (kexec)

Kernel execution templates for PXE-less boot methods.

user_data

Post-configuration scripts for providers that accept user data, such as `cloud-init` scripts.

Script

An arbitrary script not used by default but useful for custom tasks.

ZTP

Zero Touch Provisioning templates.

POAP

PowerOn Auto Provisioning templates.

iPXE

Templates for **`iPXE`** or **`gPXE`** environments to use instead of PXELinux.

3.6.2. Template Syntax and Management

Red Hat Satellite includes many template examples. In the Satellite web UI, navigate to **Hosts** > **Provisioning templates** to view them. You can create a template or clone a template and edit the clone. For help with templates, navigate to **Hosts** > **Provisioning templates** > **Create Template** > **Help**.

Templates accept the Embedded Ruby (ERB) syntax. For more information, see Template Writing Reference in *Managing Hosts*.

You can download provisioning templates. Before you can download the template, you must create a debug certificate. For more information, see Creating an Organization Debug Certificate in the *Content Management Guide*.

You can synchronize templates between Satellite Server and a Git repository or a local directory. For more information, see Appendix F. Synchronizing Templates with Git in the *Content Management* guide.

Change logs and history

To view the history of changes applied to a template, navigate to **Hosts** > **Provisioning templates**, select one of the templates, and click **History**. Click **Revert** to override the editor content with the previous version. It is possible to revert to an earlier change as well. Click **Show Diff** to see information about a specific change:

1. **Template Diff** tab displays changes in the body of a provisioning template.
2. **Details** tab displays changes in the template description.
3. **History** tab displays the user who made a change to the template and date of the change.

3.6.3. Procedure

To create a template, complete the following step:

- In the Satellite web UI, navigate to **Hosts** > **Provisioning Templates** and, in the Provisioning Templates window, click **Create Template**.

The **Help** tab provides information about the template syntax. It details the available functions, variables, and methods that can be called on different types of objects within the template.

Alternatively, to clone a template and add your updates to the clone, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Provisioning Templates** and search for the template that you want to use.
2. On the template that you want to use, click **Clone** to duplicate it.
3. In the **Name** field, enter a name for the provisioning template.
4. Select the **Default** check box to set the template to associate automatically with new organizations or locations.
5. In the **Template editor** field, enter the body of the provisioning template. You can also use the **Template** file browser to upload a template file.
6. In the **Audit Comment** field, enter a summary of changes to the provisioning template for auditing purposes.
7. Click the **Type** tab and if your template is a snippet, select the **Snippet** check box. A snippet is not a standalone provisioning template, but a part of a provisioning template that can be inserted into other provisioning templates.
8. From the **Type** list, select the type of the template. For example, **Provisioning template**.
9. Click the **Association** tab and from the **Applicable Operating Systems** list, select the names of the operating systems that you want to associate with the provisioning template.
10. Optionally, click **Add combination** and select a host group from the **Host Group** list or an environment from the **Environment** list to associate provisioning template with the host groups and environments.
11. Click the **Organizations** and **Locations** tabs to add any additional contexts to the template.
12. Click **Submit** to save your provisioning template.

For CLI Users

Before you create a template with the CLI, create a plain text file that contains the template. This example uses the **~/my-template** file. Create the installation medium using the **hammer template create** command and specify the type with the **--type** option:

```
# hammer template create --name "My Provisioning Template" \
--file ~/my-template --type provision --organizations
"My_Organization" \
--locations "My_Location"
```

3.7. CREATING COMPUTE PROFILES

Compute profiles are used in conjunction with compute resources, such as virtualization infrastructure and cloud providers. Compute profiles allow users to predefine hardware such as CPUs, memory, and storage. A default installation of Red Hat Satellite 6 contains three predefined profiles:

- **1-Small**
- **2-Medium**
- **3-Large**

Procedure

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Profiles**, and in the Compute Profiles window, click **Create Compute Profile**.
2. In the **Name** field, enter a name for the profile and click **Submit**.

For CLI Users

The compute profile CLI commands are not yet implemented in Red Hat Satellite 6.4.

3.8. SETTING A DEFAULT ENCRYPTED ROOT PASSWORD

FOR HOSTS

If you do not want to set a plain text default root password for the hosts that you provision, you can use a default encrypted password.

To set a default encrypted password for your hosts, complete the following steps:

1. Generate an encrypted password. You can use the following command to generate a password:

   ```
   # python -c 'import crypt,getpass;pw=getpass.getpass();
   print(crypt.crypt(pw)) if (pw==getpass.getpass("Confirm: "))
   else exit()'
   ```

2. Copy the password for later use.
3. In the Satellite web UI, navigate to **Administer** > **Settings**.
4. On the **Settings** page, select the **Provisioning** tab.
5. In the **Name** column, navigate to **Root password**, and click **Click to edit**.
6. Paste the encrypted password that you generate, and click **Save**.

3.9. USING THIRD PARTY INSTALLATION MEDIA

Installation media are sources of files for third parties that Satellite Server uses to install a third-party base operating system on a machine. Installation media must be in the format of an operating system installation tree, and must be accessible to the machine hosting the installer through an HTTP URL. You can view installation media by navigating to **Hosts** > **Installation Media** menu.

For other installation media, for example, a locally mounted ISO image, you can add your own custom media paths using the following procedure.

Procedure

To create installation media, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Installation Media** and click **Create Medium**.

2. In the **Name** field, enter a name to represent the installation media entry.

3. In the **Path** enter the URL or NFS share that contains the installation tree. You can use following variables in the path to represent multiple different system architectures and versions:

   - **`$arch`** - The system architecture.
   - **`$version`** - The operating system version.
   - **`$major`** - The operating system major version.
   - **`$minor`** - The operating system minor version.

   Example HTTP path:

   ```
   http://download.example.com/centos/$version/Server/$arch/os/
   ```

   Example NFS path:

   ```
   nfs://download.example.com:/centos/$version/Server/$arch/os/
   ```

   Synchronized content on Capsule Servers always uses an HTTP path. Capsule Server managed content does not support NFS paths.

4. From the **Operating system family** list, select the distribution or family of the installation medium. For example, CentOS, and Fedora are in the **`Red Hat`** family.

5. Click the **Organizations** and **Locations** tabs, to change the provisioning context. Satellite Server adds the installation medium to the set provisioning context.

6. Click **Submit** to save your installation medium.

For CLI Users

Create the installation medium using the **`hammer medium create`** command:

```
# hammer medium create --name "CustomOS" --os-family "Redhat" \
```

```
--path http://download.example.com/centos/$version/Server/$arch/os/
\
--organizations "My_Organization" --locations "My_Location"
```

3.10. USING NOVNC TO ACCESS VIRTUAL MACHINES

You can use your browser to access the VNC console of VMs created by Satellite.

Satellite supports using noVNC on the following virtualization platforms:

- VMware
- Libvirt
- RHV

Use the following procedure to configure your virtualization platform and browser to enable the use of the noVNC console.

Prerequisites

- You must have a virtual machine created by Satellite.
- For existing virtual machines, ensure that the **Display type** in the **Compute Resource** settings is **VNC**.
- You must import the Katello root CA certificate into your Satellite Server. Adding a security exception in the browser is not enough for using noVNC. For more information, see the Installing the Katello Root CA Certificate section in the *Administering Red Hat Satellite* guide.

Procedure

1. On the VM host system, configure the firewall to allow VNC service on ports 5900 to 5930:
   - On Red Hat Enterprise Linux 6:

     ```
     # iptables -A INPUT -p tcp --dport 5900:5930 -j ACCEPT
     # service iptables save
     ```

- On Red Hat Enterprise Linux 7:

```
# firewall-cmd --add-port=5900-5930/tcp
# firewall-cmd --add-port=5900-5930/tcp --permanent
```

2. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and select the name of a compute resource.
3. In the **Virtual Machines** tab, select the name of a VM host. Ensure the machine is powered on and then select **Console**.

CHAPTER 4. CONFIGURING NETWORKING

Each provisioning type requires some network configuration. Use this chapter to configure network services in Satellite Server's integrated Capsule.

New hosts must have access to your Capsule Server. A Capsule Server can be either Satellite Server's integrated Capsule or an external Capsule Server. You might want to provision hosts from an external Capsule Server when the hosts are on isolated networks and cannot connect to the Satellite Server directly, or when the content is synchronized with the Capsule Server. Provisioning using the external Capsule Server can save on network bandwidth.

Configuring the Capsule Server has two basic requirements:

1. Configuring network services. This includes:
   - Content delivery services
   - Network services (DHCP, DNS, and TFTP)
   - Puppet configuration

2. Defining network resource data in Satellite Server to help configure network interfaces on new hosts.

The following instructions have similar applications to configuring standalone Capsule Servers managing a specific network. To configure Satellite to use external DHCP, DNS, and TFTP services, see Configuring External Services in *Installing Satellite Server from a Connected Network*.

DHCP Ranges

You can define the same DHCP range in Satellite Server for both Discovered and Provisioned systems, but use a separate range for each service within the same subnet.

4.1. PREREQUISITES FOR IMAGE BASED PROVISIONING

Post-Boot Configuration Method

Images that use the **`finish`** post-boot configuration scripts require a managed DHCP server, such as Satellite's integrated Capsule or an external Capsule. The host must be created with a subnet associated with a DHCP Capsule, and the IP address of the host must be a valid IP address from the DHCP range.

It is possible to use an external DHCP service, but IP addresses must be entered manually. The SSH credentials corresponding to the configuration in the image must be configured in Satellite to enable the post-boot configuration to be made.

Check following items when troubleshooting a virtual machine booted from an image that depends on post-configuration scripts:

- The host has a subnet assigned in Satellite Server.
- The subnet has a DHCP Capsule assigned in Satellite Server.
- The host has a valid IP address assigned in Satellite Server.
- The IP address acquired by the virtual machine using DHCP matches the address configured in Satellite Server.
- The virtual machine created from an image responds to SSH requests.
- The virtual machine created from an image authorizes the user and password, over SSH, which is associated

with the image being deployed.

- Satellite Server has access to the virtual machine via SSH keys. This is required for the virtual machine to receive post-configuration scripts from Satellite Server.

Pre-Boot Initialization Configuration Method

Images that use the **`cloud-init`** scripts require a DHCP server to avoid having to include the IP address in the image. A managed DHCP Capsule is preferred. The image must have the **`cloud-init`** service configured to start when the system boots and fetch a script or configuration data to use in completing the configuration.

Check the following items when troubleshooting a virtual machine booted from an image that depends on initialization scripts included in the image:

- There is a DHCP server on the subnet.
- The virtual machine has the **`cloud-init`** service installed and enabled.

For information about the differing levels of support for **`finish`** and **`cloud-init`** scripts in virtual-machine images, see the Red Hat Knowledgebase Solution What are the supported compute resources for the finish and cloud-init scripts on the Red Hat Customer Portal.

4.2. CONFIGURING NETWORK SERVICES

Some provisioning methods use Capsule Server services. For example, a network might require the Capsule Server to act as a DHCP server. A network can also use PXE boot services to install the operating system on new hosts. This requires configuring the Capsule Server to use the main PXE boot services: DHCP, DNS, and TFTP.

Use the **`satellite-installer`** script with the options to configure these services on the Satellite Server.

To configure these services on an external Capsule Server, run **`satellite-installer --scenario capsule`**.

Satellite Server uses **`eth0`** for external communication, such as connecting to Red Hat’s CDN.

Procedure

To configure network services on Satellite’s integrated Capsule, complete the following steps:

1. Enter the **`satellite-installer`** command to configure the required network services:

```
# satellite-installer --foreman-proxy-dhcp true \
--foreman-proxy-dhcp-managed true \
--foreman-proxy-dhcp-gateway "192.168.140.1" \
--foreman-proxy-dhcp-interface "eth1" \
--foreman-proxy-dhcp-nameservers "192.168.140.2" \
--foreman-proxy-dhcp-range "192.168.140.10 192.168.140.110" \
--foreman-proxy-dhcp-server "192.168.140.2" \
--foreman-proxy-dns true \
--foreman-proxy-dns-managed true \
--foreman-proxy-dns-forwarders "8.8.8.8; 4.4.4.4" \
--foreman-proxy-dns-interface "eth1" \
--foreman-proxy-dns-reverse "140.168.192.in-addr.arpa" \
--foreman-proxy-dns-server "192.168.140.2" \
--foreman-proxy-dns-zone "example.com" \
--foreman-proxy-tftp true \
--foreman-proxy-tftp-managed true
```

2. Find the Capsule Server that you configure:

```
# hammer proxy list
```

3. Refresh features of the Capsule Server to view the changes:

```
# hammer proxy refresh-features --name "satellite.example.com"
```

4. Verify the services configured on the Capsule Server:

```
# hammer proxy info --name "satellite.example.com"
```

4.2.1. DHCP, DNS, and TFTP Options for Network Configuration

DHCP Options

--foreman-proxy-dhcp

Enables the DHCP service. You can set this option to **`true`** or **`false`**.

--foreman-proxy-dhcp-managed

Enables Foreman to manage the DHCP service. You can set this option to **`true`** or **`false`**.

--foreman-proxy-dhcp-gateway

The DHCP pool gateway. Set this to the address of the external gateway for hosts on your private network.

--foreman-proxy-dhcp-interface

Sets the interface for the DHCP service to listen for requests. Set this to **`eth1`**.

--foreman-proxy-dhcp-nameservers

Sets the addresses of the nameservers provided to clients through DHCP. Set this to the address for Satellite Server on **`eth1`**.

--foreman-proxy-dhcp-range

A space-separated DHCP pool range for Discovered and Unmanaged services.

--foreman-proxy-dhcp-server

Sets the address of the DHCP server to manage.

DNS Options

--foreman-proxy-dns

Enables DNS service. You can set this option to **`true`** or **`false`**.

--foreman-proxy-dns-managed

Enables Foreman to manage the DNS service. You can set this option to **`true`** or **`false`**.

--foreman-proxy-dns-forwarders

Sets the DNS forwarders. Set this to your DNS servers.

--foreman-proxy-dns-interface

Sets the interface to listen for DNS requests. Set this to `eth1`.

--foreman-proxy-dns-reverse

The DNS reverse zone name.

--foreman-proxy-dns-server

Sets the address of the DNS server to manage.

--foreman-proxy-dns-zone

Sets the DNS zone name.

TFTP Options

--foreman-proxy-tftp

Enables TFTP service. You can set this option to `true` or `false`.

--foreman-proxy-tftp-managed

Enables Foreman to manage the TFTP service. You can set this option to `true` or `false`.

Run `satellite-installer --scenario capsule --help` to view more options related to DHCP, DNS, TFTP, and other Satellite Capsule services

4.2.2. Using TFTP Services through NAT

You can use Satellite TFTP services through NAT. To do this, on all NAT routers or firewalls, you must enable a TFTP service on UDP port 69 and enable the TFTP state tracking feature. For more information, see the documentation for your NAT device.

If your NAT routers or firewalls use Red Hat Enterprise Linux, perform this procedure on all devices.

On Red Hat Enterprise Linux 7:

Use the following command to allow TFTP service on UDP port 69, load the kernel TFTP state tracking module, and make the changes persistent:

```
# firewall-cmd --add-service=tftp && firewall-cmd --runtime-
to-permanent
```

On Red Hat Enterprise Linux 6:

1. Configure the firewall to allow TFTP service UDP on port 69.

   ```
   # iptables -A OUTPUT -i eth0 -p udp --sport 69 -m state \
   --state ESTABLISHED -j ACCEPT
   # service iptables save
   ```

2. Load the **`ip_conntrack_tftp`** kernel TFTP state module. In the **`/etc/sysconfig/iptables-config`** file, locate **`IPTABLES_MODULES`** and add **`ip_conntrack_tftp`** as follows:

   ```
   IPTABLES_MODULES="ip_conntrack_tftp"
   ```

4.3. ADDING A DOMAIN TO SATELLITE SERVER

Satellite Server defines domain names for each host on the network. Satellite Server must have information about the domain and the Capsule Server responsible for domain name assignment.

Checking for Existing Domains

Satellite Server might already have the relevant domain created as part of Satellite Server installation. Switch the context to **`Any Organization`** and **`Any Location`** then check the domain list to see if it exists.

Procedure

To add a domain to Satellite, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Domains** and click **Create Domain**.
2. In the **DNS Domain** field, enter the full DNS domain name.
3. In the **Fullname** field, enter the plain text name of the domain.
4. Click the **Parameters** tab and configure any domain level parameters to apply to hosts attached to this domain. For example, user defined Boolean or string parameters to use in templates.
5. Click **Add Parameter** and fill in the **Name** and **Value** fields.
6. Click the **Locations** tab, and add the location where the domain resides.
7. Click the **Organizations** tab, and add the organization that the domain belongs to.
8. Click **Submit** to save the changes.

For CLI Users

Use the **`hammer domain create`** command to create a domain:

```
# hammer domain create --name "domain_name.com" \
--description "My example domain" --dns-id 1 \
--locations "My_Location" --organizations "My_Organization"
```

In this example, the **`--dns-id`** option uses **`1`**, which is the ID of Satellite Server’s integrated Capsule.

4.4. ADDING A SUBNET TO SATELLITE SERVER

You must add information for each of your subnets to Satellite Server because Satellite configures interfaces for new hosts. To configure interfaces, Satellite Server must have all the information about the network that connects these interfaces.

Procedure

To add a subnet to Satellite Server, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Subnets**, and in the Subnets window, click **Create Subnet**.
2. In the **Name** field, enter a name for the subnet.
3. In the **Description** field, enter a description for the subnet.
4. In the **Network address** field, enter the network address for the subnet.
5. In the **Network prefix** field, enter the network prefix for the subnet.
6. In the **Network mask** field, enter the network mask for the subnet.
7. In the **Gateway address** field, enter the external gateway for the subnet.
8. In the **Primary DNS server** field, enter a primary DNS for the subnet.
9. In the **Secondary DNS server**, enter a secondary DNS for the subnet.
10. From the **IPAM** list, select the method that you want to use for IP address management (IPAM):
    - **DHCP -** The subnet contains a DHCP server.
    - **Internal DB -** The subnet does not contain a DHCP server but Satellite can manage the IP address assignment and record IP addresses in its internal database.
    - **None -** No IP address management.
11. Enter the information for the IPAM method that you select.
12. Click the **Remote Execution** tab and select the capsule that controls the remote execution.
13. Click the **Domains** tab and select the domains that apply to this subnet.
14. Click the **Capsules** tab and select the capsule that applies to each service in the subnet, including DHCP, TFTP, and reverse DNS services.
15. Click the **Parameters** tab and configure any subnet level parameters to apply to hosts attached to this subnet. For example, user defined Boolean or string parameters to use in templates.
16. Click the **Locations** tab and select the locations that use this capsule.
17. Click the **Organizations** tab and select the organizations that use this capsule.

18. Click **Submit** to save the subnet information.

For CLI Users

Create the subnet with the following command:

```
# hammer subnet create --name "My_Network" \
--description "your_description" \
--network "192.168.140.0" --mask "255.255.255.0" \
--gateway "192.168.140.1" --dns-primary "192.168.140.2" \
--dns-secondary "8.8.8.8" --ipam "DHCP" \
--from "192.168.140.111" --to "192.168.140.250" --boot-mode "DHCP"
\
--domains "example.com" --dhcp-id 1 --dns-id 1 --tftp-id 1 \
--locations "My_Location" --organizations "My_Organization"
```

Note

In this example, the **`--dhcp-id`**, **`--dns-id`**, and **`--tftp-id`** options use 1, which is the ID of the integrated Capsule in Satellite Server.

4.5. CONFIGURING IPXE TO REDUCE PROVISIONING TIMES

In Red Hat Satellite 6.4, you can configure PXELinux to chainboot iPXE and boot using the HTTP protocol, which is faster and more reliable on high latency networks than TFTP.

There are three methods of using iPXE with Red Hat Satellite 6.4:

1. Chainbooting virtual machines using hypervisors that use iPXE as primary firmware
2. Using PXELinux through TFTP to chainload iPXE directly on bare metal hosts

3. Using PXELinux through UNDI, which uses HTTP to transfer the kernel and the initial RAM disk on bare metal hosts

Prerequisites

Before you begin, ensure that the following conditions are met:

- A host exists on Red Hat Satellite to use
- The MAC address of the provisioning interface matches the host configuration
- The provisioning interface of the host has a valid DHCP reservation
- The NIC is capable of PXE booting. For more information, see http://ipxe.org/appnote/hardware_drivers
- The NIC is compatible with iPXE

4.5.1. Chainbooting virtual machines

Most virtualization hypervisors use iPXE as primary firmware for PXE booting. Because of this, you can chainboot without TFTP and PXELinux.

Chainbooting virtual machine workflow

Using virtualization hypervisors removes the need for TFTP and PXELinux. It has the following workflow:

1. Virtual machine starts
2. iPXE retrieves the network credentials using DHCP
3. iPXE retrieves the HTTP address using DHCP
4. iPXE chainloads the iPXE template from the template Capsule
5. iPXE loads the kernel and initial RAM disk of the installer

Ensure that the hypervisor that you want to use supports iPXE. The following virtualization hypervisors support iPXE:

- libvirt
- oVirt
- RHEV

Configuring Red Hat Satellite Server to use iPXE

You can use the default template to configure iPXE booting for hosts. If you want to change the default values in the template, clone the template and edit the clone.

Procedure

To configure Satellite to use iPXE, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Provisioning Templates**, enter `Kickstart default iPXE` and click **Search**.
2. Optional: If you want to change the template, click **Clone**, enter a unique name, and click **Submit**.
3. Click the name of the template you want to use.
4. If you clone the template, you can make changes you require on the **Template** tab.
5. Click the **Association** tab, and select the operating systems that your host uses.
6. Click the **Locations** tab, and add the location where the host resides.
7. Click the **Organizations** tab, and add the organization that the host belongs to.
8. Click **Submit** to save the changes.
9. Navigate to **Hosts** > **Operating systems** and select the operating system of your host.
10. Click the **Templates** tab.
11. From the **iPXE Template** list, select the template you want to use.
12. Click **Submit** to save the changes.
13. Navigate to **Hosts** > **All Hosts**.

14. In the **Hosts** page, select the host that you want to use.

15. Select the **Templates** tab.

16. From the **iPXE template** list, select **Review** and verify that the **Kickstart default iPXE** template is the correct template.

17. To prevent an endless loop of chainbooting iPXE firmware, edit the **`/etc/dhcp/dhcpd.conf`** file to match the following example. If you use an isolated network, use a Capsule Server URL with TCP port 8000, instead of Satellite Server's URL.

    a. Locate the following lines in the Bootfile Handoff section of the **`/etc/dhcp/dhcpd.conf`** file:

    ```
    } else {
      filename "pxelinux.0";
    }
    ```

    b. Add the following extra **`elsif`** statement before the else statement:

    ```
    elsif exists user-class and option user-class = "iPXE" {
      filename "http://satellite.example.com/unattended/iPXE";
    }
    ```

    c. Verify that the *if* section matches the following example:

    ```
    if option architecture = 00:06 {
      filename "grub2/shim.efi";
    } elsif option architecture = 00:07 {
      filename "grub2/shim.efi";
    } elsif option architecture = 00:09 {
      filename "grub2/shim.efi";
    } elsif exists user-class and option user-class = "iPXE" {
      filename "http://satellite.example.com/unattended/iPXE";
    } else {
      filename "pxelinux.0";
    ```

```
}
```

Note

For **`http://satellite.example.com/unattended/iPXE`**, you can also use a Red Hat Satellite Capsule **`http://capsule.example.com:8000/unattended/iPXE`**. You must update the **`/etc/dhcp/dhcpd.conf`** file after every upgrade. The content of the **`/etc/dhcp/dhcpd.conf`** file is case sensitive.

4.5.2. Chainbooting iPXE directly

Use this procedure to set up iPXE to use a built-in driver for network communication or UNDI interface. There are separate procedures to configure Satellite Server and Capsule to use iPXE.

You can use this procedure only with bare metal hosts.

Chainbooting iPXE directly or with UNDI workflow

1. Host powers on
2. PXE driver retrieves the network credentials using DHCP
3. PXE driver retrieves the PXELinux firmware **`pxelinux.0`** using TFTP
4. PXELinux searches for the configuration file on the TFTP server
5. PXELinux chainloads iPXE **`ipxe.lkrn`** or **`undionly-ipxe.0`**
6. iPXE retrieves the network credentials using DHCP again
7. iPXE retrieves HTTP address using DHCP
8. iPXE chainloads the iPXE template from the template Capsule
9. iPXE loads the kernel and initial RAM disk of the installer

Configuring Red Hat Satellite Server to use iPXE

You can use the default template to configure iPXE booting for hosts. If you want to change the default values in the template, clone the template and edit the clone.

Procedure

To configure Satellite to use iPXE with the UNDI workflow, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Provisioning Templates**, enter `PXELinux chain iPXE` or, for UNDI, enter `PXELinux chain iPXE UNDI`, and click **Search**.
2. Optional: If you want to change the template, click **Clone**, enter a unique name, and click **Submit**.
3. Click the name of the template you want to use.
4. If you clone the template, you can make changes you require on the **Template** tab.
5. Click the **Association** tab, and select the operating systems that your host uses.
6. Click the **Locations** tab, and add the location where the host resides.
7. Click the **Organizations** tab, and add the organization that the host belongs to.
8. Click **Submit** to save the changes.
9. In the **Provisioning Templates** page, enter `Kickstart default iPXE` into the search field and click **Search**.
10. Optional: If you want to change the template, click **Clone**, enter a unique name, and click **Submit**.
11. Click the name of the template you want to use.
12. If you clone the template, you can make changes you require on the **Template** tab.
13. Click the **Association** tab, and associate the template with the operating system that your host uses.
14. Click the **Locations** tab, and add the location where the host resides.
15. Click the **Organizations** tab, and add the organization that the host belongs to.
16. Click **Submit** to save the changes.

17. Navigate to **Hosts** > **Operating systems** and select the operating system of your host.

18. Click the **Templates** tab.

19. From the **PXELinux template** list, select the template you want to use.

20. From the **iPXE template** list, select the template you want to use.

21. Click **Submit** to save the changes.

22. Navigate to **Hosts** > **All Hosts**, and select the host you want to use.

23. Select the **Templates** tab, and from the **PXELinux template** list, select **Review** and verify the template is the correct template.

24. From the **iPXE template** list, select **Review** and verify the template is the correct template. If there is no PXELinux entry, or you cannot find the new template, navigate to **Hosts** > **All Hosts**, and on your host, click **Edit**. Click the **Operating system** tab and click the Provisioning Template **Resolve** button to refresh the list of templates.

25. To prevent an endless loop of chainbooting iPXE firmware, edit the **`/etc/dhcp/dhcpd.conf`** file to match the following example. If you use an isolated network, use a Capsule Server URL with TCP port 8000, instead of Satellite Server’s URL.

    a. Locate the following lines in the Bootfile Handoff section of the **`/etc/dhcp/dhcpd.conf`** file:

    ```
    } else {
      filename "pxelinux.0";
    }
    ```

    b. Add the following extra **`elsif`** statement before the else statement:

    ```
    elsif exists user-class and option user-class = "iPXE" {
      filename "http://satellite.example.com/unattended/iPXE";
    }
    ```

    c. Verify that the *if* section matches the following example:

```
if option architecture = 00:06 {
  filename "grub2/shim.efi";
} elsif option architecture = 00:07 {
  filename "grub2/shim.efi";
} elsif option architecture = 00:09 {
  filename "grub2/shim.efi";
} elsif exists user-class and option user-class = "iPXE" {
  filename "http://satellite.example.com/unattended/iPXE";
} else {
  filename "pxelinux.0";
}
```

Note

For **`http://satellite.example.com/unattended/iPXE`**, you can also use a Red Hat Satellite Capsule **`http://capsule.example.comf:8000/unattended/iPXE`**. You must update the **`/etc/dhcp/dhcpd.conf`** file after every upgrade. The content of the **`/etc/dhcp/dhcpd.conf`** file is case sensitive.

Configuring Red Hat Satellite Capsule to use iPXE

You can use this procedure to configure Capsules to use iPXE.

You must perform this procedure on all Capsules.

Procedure

To configure the Capsule to chainboot iPXE, complete the following steps:

1. Install the **`ipxe-bootimgs`** RPM package:

   ```
   # yum install ipxe-bootimgs
   ```

2. Copy the iPXE firmware to the TFTP server’s root directory. Do not use symbolic links because TFTP runs in

the **chroot** environment.

- For chainbooting directly, enter the following command:

```
# cp /usr/share/ipxe/ipxe.lkrn /var/lib/tftpboot/
```

- For UNDI, enter the following command:

```
# cp /usr/share/ipxe/undionly.kpxe /var/lib/tftpboot
/undionly-ipxe.0
```

3. Correct the file contexts:

```
# restorecon -RvF /var/lib/tftpboot/
```

CHAPTER 5. PROVISIONING BARE METAL HOSTS

There are four main ways to provision bare metal instances with Red Hat Satellite 6.4:

Unattended Provisioning

New hosts are identified by a MAC address and Satellite Server provisions the host using a PXE boot process.

Unattended Provisioning with Discovery

New hosts use PXE boot to load the Satellite Discovery service. This service identifies hardware information about the host and lists it as an available host to provision.

PXE-less Provisioning

New hosts are provisioned with a boot disk or PXE-less discovery image that Satellite Server generates.

PXE-less Provisioning with Discovery

New hosts use an ISO boot disk that loads the Satellite Discovery service. This service identifies hardware information about the host and lists it as an available host to provision.

BIOS and UEFI Support

With Red Hat Satellite, you can perform both BIOS and UEFI based PXELinux provisioning.

Both BIOS and UEFI interfaces work as interpreters between the computer’s operating system and firmware, initializing the hardware components and starting the operating system at boot time.

While BIOS reads the first section of the hard drive that contains the next address to initialize, UEFI stores all the information about initialization and startup in an **`.efi`** file instead of the firmware. UEFI systems are newer and becoming more common.

To perform PXELinux provisioning with UEFI you must use a Red Hat Enterprise Linux Server 7 or higher that has Intel x86_64. In Satellite, PXELinux provisioning with UEFI is supported only on bare-metal systems. UEFI is not supported for virtual machines. UEFI SecureBoot is also not supported.

In Satellite provisioning, the PXE loader which is a DHCP file name that defines which file to load through TFTP during PXE provisioning. For BIOS system, the file is **`pxelinux.0`**. For UEFI systems, the file is **`grub2/grubx64.efi`**.

For BIOS provisioning, you must associate a PXELinux template with the operating system.

For UEFI provisioning you must associate a PXEGrub2 template with the operating system.

If you associate both PXELinux and PXEGrub2 templates, Satellite 6 can deploy configuration files for both on a TFTP server, so that you can switch between PXE loaders easily.

5.1. PREREQUISITES FOR BARE METAL PROVISIONING

The requirements for bare metal provisioning include:

- Synchronized content repositories for Red Hat Enterprise Linux 7. For more information, see Synchronizing Red Hat Repositories in the *Content Management Guide*.

- A Capsule Server managing the network for bare metal hosts. For unattended provisioning and discovery-based provisioning, Satellite Server requires PXE server settings. For more information, see Chapter 4, *Configuring Networking*.
- An activation key for host registration. For more information, see Creating An Activation Key in the *Content Management* guide.
- A blank bare metal host.

For information about the security token for unattended and PXE-less provisioning, see Section 5.2, “Configuring the Security Token Validity Duration”.

5.2. CONFIGURING THE SECURITY TOKEN VALIDITY DURATION

When performing unattended and PXE-less provisioning, as a security measure, Satellite automatically generates a unique token and adds this token to the URL of the ISO image that downloads during the Kickstart provisioning process.

By default, the token is valid for 360 minutes. When you provision a host, ensure that you reboot the host within this time frame. If the token expires, it is deleted and you might receive a 404 error.

To adjust the token’s duration of validity, in the Satellite web UI, navigate to **Administer** > **Settings**, and click the **Provisioning** tab. Find the **Token duration** option, and click the edit icon and edit the duration, or enter `0` to disable token generation.

5.3. CREATING HOSTS WITH UNATTENDED PROVISIONING

Unattended provisioning is the simplest form of host provisioning. You enter the host details on Satellite Server and boot your host. Satellite Server automatically manages the PXE configuration, organizes networking services, and provides the operating system and configuration for the host.

This method of provisioning hosts uses minimal interaction during the process.

Procedure

To create a host with unattended provisioning, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Create Host**.
2. In the **Name** field, enter a name for the host.
3. Click the **Organization** and **Location** tabs and change the context to match your requirements.
4. From the **Host Group** list, select a host group that you want to use to populate the form.
5. Click the **Interface** tab, and on the host's interface, click **Edit**.
6. Verify that the fields are populated with values. Note in particular:
   - The **Name** from the **Host** tab becomes the **DNS name**.
   - Satellite Server automatically assigns an IP address for the new host.
7. In the **MAC address** field, enter a MAC address for the host. This ensures the identification of the host during the PXE boot process.
8. Ensure that Satellite Server automatically selects the **Managed**, **Primary**, and **Provision** options for the first interface on the host. If not, select them.
9. Click the **Operating System** tab, and verify that all fields contain values. Confirm each aspect of the operating system.
10. Click **Resolve** in **Provisioning template** to check the new host can identify the right provisioning templates to use.
11. Optional: If you want to use VLAN tagging, you must add the VLAN ID to the PXELinux or PXEGrub2 template. To the **`APPEND`** line, add **`vlanid=example_vlanid`**.

    For more information about associating provisioning templates, see Section 3.6, “Creating Provisioning Templates”.
12. Click the **Parameters** tab, and ensure that a parameter exists that provides an activation key. If not, add an

activation key.

13. Click **Submit** to save the host details.

This creates the host entry and the relevant provisioning settings. This also includes creating the necessary directories and files for PXE booting the bare metal host. If you start the physical host and set its boot mode to PXE, the host detects the DHCP service of Satellite Server's integrated Capsule and starts installing the operating system from its Kickstart tree. When the installation completes, the host also registers to Satellite Server using the activation key and installs the necessary configuration and management tools from the Red Hat Satellite Tools repository.

For CLI Users

Create the host with the **`hammer host create`** command.

```
# hammer host create --name "My_Unattended_Host" --organization
"My_Organization" \
--location "My_Location" --hostgroup "My_Host_Group" --mac
"aa:aa:aa:aa:aa:aa" \
--build true --enabled true --managed true
```

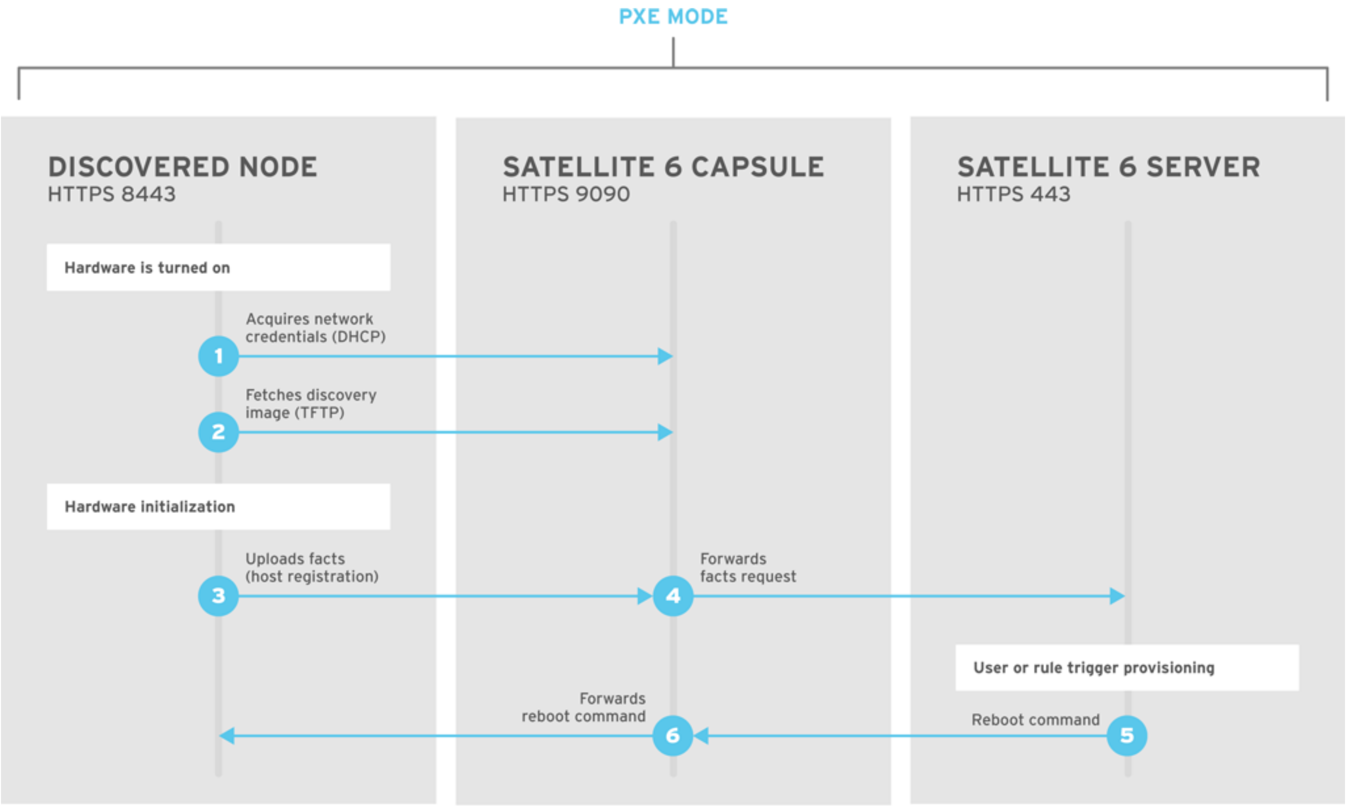
Ensure the network interface options are set using the **`hammer host interface update`** command.

```
# hammer host interface update --host "test1" --managed true \
--primary true --provision true
```

5.4. CONFIGURING RED HAT SATELLITE'S DISCOVERY SERVICE

Red Hat Satellite provides a method to automatically detect blank hosts on a network. These hosts boot the discovery image that performs hardware detection and relays this information back to Satellite Server. This method creates a list of ready-to-provision hosts in Satellite Server without needing to enter the MAC address of each host.

The Discovery service is enabled by default on Satellite Server.



SATELLITE6_376339_1115

To use Satellite Server to provide the Discovery image, install the **`foreman-discovery-image`** and **`rubygem-smart_proxy_discovery`** packages:

```
# yum install foreman-discovery-image rubygem-smart_proxy_discovery
```

The **`foreman-discovery-image`** package installs the Discovery ISO to the **`/usr/share/foreman-discovery-image/`** directory and also creates a PXE boot image from this ISO using the **`livecd-iso-to-pxeboot`** tool. The tool saves this PXE boot image in the **`/var/lib/tftpboot/boot`** directory. The **`rubygem-smart_proxy_discovery`** package configures a Capsule Server, such as Satellite Server's integrated Capsule, to act as a proxy for the Discovery service.

When the installation completes, you can view the new menu option by navigating to **Hosts** > **Discovered Hosts**.

5.4.1. Enabling Discovery service on a Capsule Server

Complete the following procedure to enable the Discovery service on a Capsule Server.

1. Enter the following commands on the Capsule Server:

   ```
   # yum install foreman-discovery-image rubygem-
   smart_proxy_discovery
   ```

   ```
   # foreman-maintain service restart
   ```

2. In the Satellite web UI, navigate to **Infrastructure** > **Capsule**.
3. Click the Capsule Server and select **Refresh** from the **Actions** list. Locate **Discovery** in the list of features to confirm the Discovery service is now running.

Subnets

All subnets with discoverable hosts require an appropriate Capsule Server selected to provide the Discovery service.

To check this, navigate to **Infrastructure** > **Capsules** and verify if the Capsule Server that you want to use lists the Discovery feature. If not, click **Refresh features**.

In the Satellite web UI, navigate to **Infrastructure** > **Subnets**, select a subnet, click the Capsules tab, and select the **Discovery Proxy** that you want to use. Perform this for each appropriate subnet.

5.4.2. Provisioning Template PXELinux Discovery Snippet

For BIOS provisioning, the **`PXELinux global default`** template in the **Hosts** > **Provisioning Templates** window contains the snippet **`pxelinux_discovery`**. The snippet has the following lines:

```
LABEL discovery
  MENU LABEL Foreman Discovery Image
  KERNEL boot/fdi-image/vmlinuz0
  APPEND initrd=boot/fdi-image/initrd0.img rootflags=loop
```

```
root=live:/fdi.iso rootfstype=auto ro rd.live.image acpi=force
rd.luks=0 rd.md=0 rd.dm=0 rd.lvm=0 rd.bootif=0 rd.neednet=0
nomodeset proxy.url=<%= foreman_server_url %> proxy.type=foreman
  IPAPPEND 2
```

The **`KERNEL`** and **`APPEND`** options boot the Discovery image and ramdisk. The **`APPEND`** option contains a **`proxy.url`** parameter, with the **`foreman_server_url`** macro as its argument. This macro resolves to the full URL of Satellite Server.

For UEFI provisioning, the **`PXEgrub2 global default`** template in the **Hosts** > **Provisioning Templates** window contains the snippet **`pxegrub2_discovery`**:

```
menuentry Foreman Discovery Image --id discovery {
  linuxefi boot/fdi-image/vmlinuz0 rootflags=loop
root=live:/fdi.iso rootfstype=auto ro rd.live.image acpi=force
rd.luks=0 rd.md=0 rd.dm=0 rd.lvm=0 rd.bootif=0 rd.neednet=0
nomodeset proxy.url=<%= foreman_server_url %> proxy.type=foreman
BOOTIF=01-$mac
  initrdefi boot/fdi-image/initrd0.img
}
```

To use a Capsule to proxy the discovery steps, edit **`/var/lib/tftpboot/pxelinux.cfg/default`** or **`/var/lib/tftpboot/grub2/grub.cfg`** and change the URL to the FQDN of the Capsule Server you want to use.

The global template is available on Satellite Server and all Capsules that have the TFTP feature enabled.

5.4.3. Changing Templates and Snippets

To use a template, in the Satellite web UI, navigate to **Administer** > **Settings** and click the **Provisioning** tab and set the templates that you want to use.

Templates and snippets are locked to prevent changes. If you want to edit a template or snippet, clone it, save it with a unique name, and then edit the clone.

When you change the template or a snippet it includes, the changes must be propagated to Satellite Server's default PXE template. Navigate to **Hosts** > **Provisioning Templates** and click **Build PXE Default**. This refreshes the default PXE template on Satellite Server.

You can change the `proxy.url` argument to the IP address or FQDN of another provisioning Capsule that you want to use, but remember to append the port number, 9090. For example:

```
proxy.url=https://capsule.example.com:9090
```

Rendering the Capsule's Host Name

Satellite 6 deploys the same template to all TFTP Capsules and there is no variable or macro available to render the Capsule's host name. The hard-coded `proxy.url` does not not work with two or more TFTP Capsules. As a workaround, every time you click **Build PXE Defaults**, edit the configuration file in the TFTP directory using SSH.

Setting Discovery Service as Default

For both BIOS and UEFI, to set the Discovery service as the default service that boots for blank hosts, complete the following steps:

1. In the Satellite web UI, navigate to **Administer** > **Settings** and click the **Provisioning** tab.
2. For the **Default PXE global template entry**, in the **Value** column, enter `discovery`.

Tagged VLAN Provisioning

If you want to use tagged VLAN provisioning, and you want the discovery service to send a discovery request, add the following information to the `KERNEL` option in the discovery template:

```
fdi.vlan.primary=example_VLAN_ID
```

Testing

Test the Discovery service and boot a blank bare metal host on the 192.168.140.0/24 network. A boot menu has two options:

- `local`, which boots from the hard disk

- **`discovery`**, which boots to the Discovery service

Select **`discovery`** to boot the Discovery image. After a few minutes, the Discovery image completes booting and a status screen is displayed.

In the Satellite web UI, navigate to **Hosts** > **Discovered hosts** and view the newly discovered host. The discovered hosts automatically define their host name based on their MAC address. For example, Satellite sets a discovered host with a MAC address of ab:cd:ef:12:34:56 to have **`macabcdef123456`** as the host name. You can change this host name when provisioning the host.

5.4.4. Automatic Contexts for Discovered Hosts

Satellite Server assigns organization and location to discovered hosts according to the following sequence of rules:

1. If the **`Discovery organization`** or **`Discovery location`** values are set. To set these values, navigate to **Administer** > **Settings** > **Discovered**.
2. If the **`foreman_organization`** or **`foreman_location`** facts for a host are set. To configure fact names in **Administer** > **Settings** > **Puppet** section as the **Default organization** and **Default location** fact setting.
3. If a discovered host uses a subnet defined in Satellite, the host uses the first organization and location associated with the subnet.
4. If none of the previous conditions exists, Satellite assigns the first Organization and Location ordered by name.

You can change the organization or location using the bulk actions menu of the **Discovered hosts** page. Select the discovered hosts to modify and select **Assign Organization** or **Assign Location** from the **Select Action** menu.

5.5. CREATING HOSTS FROM DISCOVERED HOSTS

Provisioning discovered hosts follows a provisioning process that is similar to PXE provisioning. The main difference is that instead of manually entering the host's MAC address, you can select the host to provision from the list of discovered hosts.

Procedure

To create a host from a discovered host, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Discovered host**. Select the host you want to use and click **Provision** to the right of the list.
2. Select from one of the two following options:
   - To provision a host from a host group, select a host group, organization, and location, and then click **Create Host**.
   - To provision a host with further customization, click **Customize Host** and enter the additional details you want to specify for the new host.
3. Verify that the fields are populated with values. Note in particular:
   - The **Name** from the **Host** tab becomes the **DNS name**.
   - Satellite Server automatically assigns an IP address for the new host.
   - Satellite Server automatically populates the MAC address from the Discovery results.
4. Ensure that Satellite Server automatically selects the **Managed**, **Primary**, and **Provision** options for the first interface on the host. If not, select them.
5. Click the **Operating System** tab, and verify that all fields contain values. Confirm each aspect of the operating system.
6. Click **Resolve** in **Provisioning template** to check the new host can identify the right provisioning templates to use.

   For more information about associating provisioning templates, see Section 3.6, "Creating Provisioning Templates".
7. Click **Submit** to save the host details.

When the host provisioning is complete, the discovered host becomes a content host. To view the host, navigate to **Hosts** > **Content Hosts**.

For CLI Users

1. Identify the discovered host to use for provisioning:

```
# hammer discovery list
```

2. Select a host and provision it using a host group. Set a new host name with the **`--new-name`** option:

```
# hammer discovery provision --name "host_name" \
--new-name "new_host_name" --organization "My_Organization" \
--location "My_Location" --hostgroup "My_Host_Group" --build
true \
--enabled true --managed true
```

This removes the host from the discovered host listing and creates a host entry with the provisioning settings. The Discovery image automatically resets the host so that it can boot to PXE. The host detects the DHCP service on Satellite Server's integrated Capsule and starts installing the operating system from its Kickstart tree. When the installation completes, the host also registers to Satellite Server using an activation key and installs the necessary configuration and management tools from the Red Hat Satellite Tools repository.

5.6. CREATING DISCOVERY RULES

As a method of automating the provisioning process for discovered hosts, Red Hat Satellite 6 provides a feature to create discovery rules. These rules define how discovered hosts automatically provision themselves, based on the assigned host group. For example, you can automatically provision hosts with a high CPU count as hypervisors. Likewise, you can provision hosts with large hard disks as storage servers.

NIC Considerations

Auto provisioning does not currently allow configuring NICs; all systems are being provisioned with the NIC configuration that was detected during discovery. However, you can set the NIC in an Anaconda kickstart, scriptlet, or using configuration management later on.

Procedure

To create a rule, complete the following steps:

1. In the Satellite web UI, navigate to **Configure** > **Discovery rules**. Select **Create Rule** and enter the following details:
2. In the **Name** field, enter a name for the rule.
3. In the **Search** field, enter the rules to determine whether to provision a host. This field provides suggestions for values you enter and allows operators for multiple rules. For example: `cpu_count > 8`.
4. From the **Host Group** list, select the host group to use as a template for this host.
5. In the **Hostname** field, enter the pattern to determine host names for multiple hosts. This uses the same ERB syntax that provisioning templates use. The host name can use the `@host` attribute for host-specific values and the `rand` function for a random number.
   - `myhost-<%= rand(99999) %>`
   - `abc-<%= @host.facts['bios_vendor'] + '-' + rand(99999).to_s %>`
   - `xyz-<%= @host.hostgroup.name %>`
   - `srv-<%= @host.discovery_rule.name %>`
   - `server-<%= @host.ip.gsub('.','-') + '-' + @host.hostgroup.subnet.name %>`

   Because the `rand()` function returns an integer that cannot be concatenated with a string, use the `to_s` function to change the integer to a string. When creating host name patterns, ensure the resulting host names are unique, do not start with numbers, and do not contain underscores or dots. A good approach is to use unique information provided by Facter, such as the MAC address, BIOS, or serial ID.
6. In the **Hosts limit** field, enter the maximum hosts you can provision with the rule. Enter `0` for unlimited.

7. In the **Priority** field, enter a number to set the precedence the rule has over other rules. Rules with lower values have a higher priority.

8. From the **Enabled** list, select whether you want to enable the rule.

9. To set a different provisioning context for the rule, click the **Organizations** and **Locations** tabs and select the contexts you want to use.

10. Click **Submit** to save your rule.

11. Navigate to **Hosts** > **Discovered Host** and select one of the following two options:

    - From the **Discovered hosts** list on the right, select **Auto-Provision** to automatically provisions a single host.

    - On the upper right of the window, click **Auto-Provision All** to automatically provisions all hosts.

For CLI Users

Create the rule with the **`hammer discovery_rule create`** command:

```
# hammer discovery_rule create --name "Hypervisor" \
--search "cpu_count  > 8" --hostgroup "My_Host_Group" \
--hostname "hypervisor-<%= rand(99999) %>" \
--hosts-limit 5 --priority 5 --enabled true
```

Automatically provision a host with the **`hammer discovery auto-provision`** command:

```
# hammer discovery auto-provision --name "macabcdef123456"
```

5.7. CREATING HOSTS WITH PXE-LESS PROVISIONING

Some hardware does not provide a PXE boot interface. Red Hat Satellite 6 provides a PXE-less discovery service that operates without PXE-based services, such as DHCP and TFTP. In Satellite, you can provision a host without PXE boot. This is also known as PXE-less provisioning and involves generating a boot ISO that hosts can use. Using this ISO, the host can connect to Satellite Server, boot the installation media, and install the operating system.

Boot ISO Types

There are four types of boot ISOs:

Host image - A boot ISO for the specific host. This image contains only the boot files that are necessary to access the installation media on Satellite Server. The user defines the subnet data in Satellite and the image is created with static networking.

Full host image - A boot ISO that contains the kernel and initial RAM disk image for the specific host. This image is useful if the host fails to chainload correctly. The provisioning template still downloads from Satellite Server.

Generic image - A boot ISO that is not associated with a specific host. The ISO sends the host's MAC address to Satellite Server, which matches it against the host entry. The image does not store IP address details, and requires access to a DHCP server on the network to bootstrap. This image is also available from the `/bootdisk/disks/generic` URL on your Satellite Server, for example, `https://satellite.example.com/bootdisk/disks/generic`.

Subnet image - A boot ISO that is similar to the generic image but is configured with the address of a Capsule Server. This image is generic to all hosts with a provisioning NIC on the same subnet.

Procedure

To create a host with PXE-less provisioning, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Create Host**.
2. In the **Name** field, enter a name that you want to become the provisioned system's host name.
3. Click the **Organization** and **Location** tabs and change the context to match your requirements.
4. From the **Host Group** list, select a host group that you want to use to populate the form.
5. Click the **Interface** tab, and on the host's interface, click **Edit**.
6. Verify that the fields are populated with values. Note in particular:
   - The **Name** from the **Host** tab becomes the **DNS name**.
   - Satellite Server automatically assigns an IP address for the new host.

7. In the **MAC address** field, enter a MAC address for the host.
8. Ensure that Satellite Server automatically selects the **Managed**, **Primary**, and **Provision** options for the first interface on the host. If not, select them.
9. Click the **Operating System** tab, and verify that all fields contain values. Confirm each aspect of the operating system.
10. Click **Resolve** in **Provisioning template** to check the new host can identify the right provisioning templates to use.

    For more information about associating provisioning templates, see Section 3.6, “Creating Provisioning Templates”.
11. Click the **Parameters** tab, and ensure that a parameter exists that provides an activation key. If not, add an activation key.
12. Click **Submit** to save the host details.

This creates a host entry and the host details page appears.

The options on the upper-right of the window are the **Boot disk** menu. From this menu, one of the following images is available for download: **Host image**, **Full host image**, **Generic image**, and **Subnet image**.

> **Note**
>
> The **Full host image** is based on SYSLINUX and works with most hardware. When using a **Host image**, **Generic image**, or **Subnet image**, see http://ipxe.org/appnote/hardware_drivers for a list of hardware drivers expected to work with an iPXE-based boot disk.

For CLI Users

Create the host with the **`hammer host create`** command.

```
# hammer host create --name "My_Bare_Metal" --organization
"My_Organization" \
--location "My_Location" --hostgroup "My_Host_Group" --mac
```

```
"aa:aa:aa:aa:aa:aa" \
--build true --enabled true --managed true
```

Ensure that your network interface options are set using the **`hammer host interface update`** command.

```
# hammer host interface update --host "test3" --managed true \
--primary true --provision true
```

Download the boot disk from Satellite Server with the **`hammer bootdisk host`** command:

- For **Host image**:

  ```
  # hammer bootdisk host --host test3.example.com
  ```

- For **Full host image**:

  ```
  # hammer bootdisk host --host test3.example.com --full true
  ```

- For **Generic image**:

  ```
  # hammer bootdisk generic
  ```

- For **Subnet image**:

  ```
  # hammer bootdisk subnet --subnet subnetName
  ```

This creates a boot ISO for your host to use.

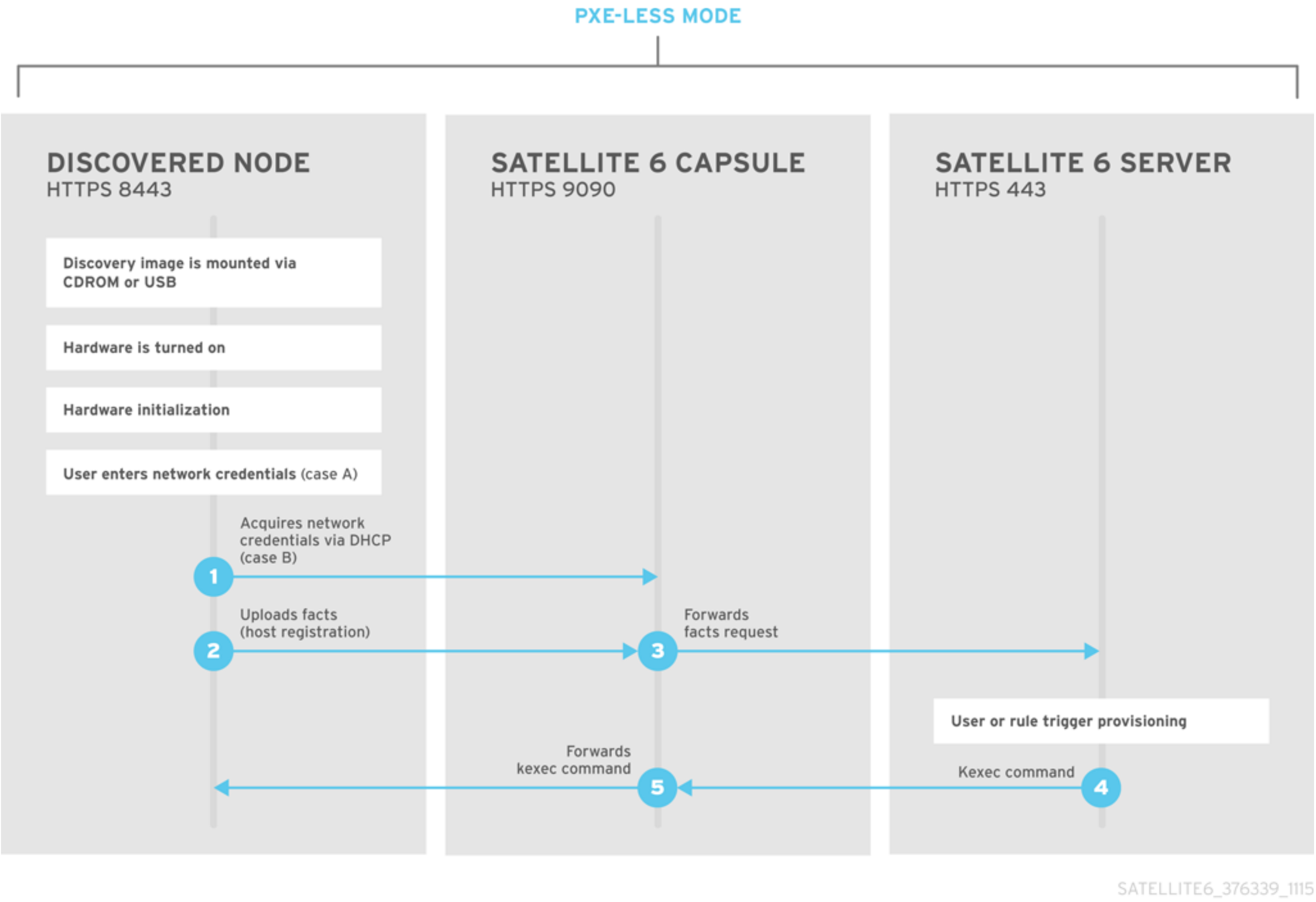
Write the ISO to a USB storage device using the **dd** utility or **livecd-tools** if required.

When you start the physical host and boot from the ISO or the USB storage device, the host connects to Satellite Server and starts installing operating system from its kickstart tree.

When the installation completes, the host also registers to Satellite Server using the activation key and installs the necessary configuration and management tools from the **Red Hat Satellite Tools** repository.

5.8. IMPLEMENTING PXE-LESS DISCOVERY

Red Hat Satellite 6 provides a PXE-less Discovery service that operates without the need for PXE-based services (DHCP and TFTP). You accomplish this using Satellite Server's Discovery image.



If you have not yet installed the Discovery service or image, follow the *"Installation"* section in Section 5.4, "Configuring Red Hat Satellite's Discovery Service".

The ISO for the Discovery service resides at **`/usr/share/foreman-discovery-image/`** and is installed using the **`foreman-discover-image`** package.

Attended Use

This ISO acts as bootable media. Copy this media to either a CD, DVD, or a USB stick. For example, to copy to a USB stick at **`/dev/sdb`**:

```
# dd bs=4M \
if=/usr/share/foreman-discovery-image/foreman-discovery-image-
3.4.4-5.iso \
of=/dev/sdb
```

Insert the Discovery boot media into a bare metal host, start the host, and boot from the media. The Discovery Image displays an option for either **Manual network setup** or **Discovery with DHCP**:

- If selecting **Manual network setup**, the Discovery image requests a set of network options. This includes the primary network interface that connects to Satellite Server. This Discovery image also asks for network interface configuration options, such as an **IPv4 Address**, **IPv4 Gateway**, and an **IPv4 DNS** server.

  After entering these details, select **Next**.

- If selecting **Discovery with DHCP**, the Discovery image requests only the primary network interface that connects to Satellite Server. It attempts to automatically configure the network interface using a DHCP server, such as one that a Capsule Server provides.

After the primary interface configuration, the Discovery image requests the **Server URL**, which is the URL of Satellite Server or Capsule Server offering the Discovery service. For example, to use the integrated Capsule on Satellite Server, use the following URL:

`https://satellite.example.com:9090`

Set the **Connection type** to **`Proxy`**, then select **Next**.

The Discovery image also provides a set of fields to input **Custom facts** for the Facter tool to relay back to Satellite Server. These are entered in a **name-value** format. Provide any custom facts you require and select **Confirm** to continue.

The Satellite reports a successful communication with Satellite Server's Discovery service. Navigate to **Hosts** > **Discovered Hosts** and view the newly discovered host.

For more information about provisioning discovered hosts, see Section 5.5, "Creating Hosts from Discovered Hosts".

Unattended Use and Customization

It is possible to create a customized Discovery ISO, which automates the process of configuring the image after booting. The Discovery image uses a Linux kernel for the operating system, which means you pass kernel parameters to the configure the image's operating system. These kernel parameters include:

proxy.url

The URL of the Capsule Server providing the Discovery service.

proxy.type

The proxy type. This is usually set to `proxy` to connect to Capsule Server. This parameter also supports a legacy `foreman` option, where communication goes directly to Satellite Server instead of a Capsule Server.

fdi.pxmac

The MAC address of the primary interface in the format of `AA:BB:CC:DD:EE:FF`. This is the interface you aim to use for communicating with Capsule Server. In automated mode, the first NIC (using network identifiers in alphabetical order) with a link is used. In semi-automated mode, a screen appears and requests you to select the correct interface.

fdi.pxip, fdi.pxgw, fdi.pxdns

Manually configures IP address (`fdi.pxip`), the gateway (`fdi.pxgw`), and the DNS (`fdi.pxdns`) for the primary network interface. If your omit these parameters, the image uses DHCP to configure the network interface.

fdi.pxfactname1, fdi.pxfactname2 … fdi.pxfactnameN

Allows you to specify custom fact names.

fdi.pxfactvalue1, fdi.pxfactvalue2 … fdi.pxfactvalueN

The values for each custom fact. Each value corresponds to a fact name. For example, `fdi.pxfactvalue1` sets the value for the fact named with `fdi.pxfactname1`.

fdi.pxauto

To set automatic or semi-automatic mode. If set to 0, the image uses semi-automatic mode, which allows you to confirm your choices through a set of dialog options. If set to 1, the image uses automatic mode and proceeds without any confirmation.

Satellite Server also provides a tool (**discovery-remaster**) in the **foreman-discovery-image** package. This tool remasters the image to include these kernel parameters. To remaster the image, run the **discovery-remaster** tool. For example:

```
# discovery-remaster ~/iso/foreman-discovery-image-3.4.4-5.iso \
"fdi.pxip=192.168.140.20/24 fdi.pxgw=192.168.140.1 \
fdi.pxdns=192.168.140.2
proxy.url=https://satellite.example.com:9090 \
proxy.type=proxy fdi.pxfactname1=customhostname \
fdi.pxfactvalue1=myhost fdi.pxmac=52:54:00:be:8e:8c fdi.pxauto=1"
```

The tool creates a new ISO file in the same directory as the original discovery image. In this scenario, it saves in the **/usr/share/foreman-discovery-image/** directory.

Copy this media to either a CD, DVD, or a USB stick. For example, to copy to a USB stick at **/dev/sdb**:

```
# dd bs=4M \
if=/usr/share/foreman-discovery-image/foreman-discovery-image-
3.4.4-5.iso \
of=/dev/sdb
```

Insert the Discovery boot media into a bare metal host, start the host, and boot from the media.

For more information about provisioning discovered hosts, see Section 5.5, “Creating Hosts from Discovered Hosts”.

Final Notes

The host needs to resolve to the following provisioning templates:

- **kexec Template: Discovery Red Hat kexec**
- **provision Template: Satellite Kickstart Default**

For more information about associating provisioning templates, see Section 3.6, “Creating Provisioning Templates”.

5.9. DEPLOYING SSH KEYS DURING PROVISIONING

Use this procedure to deploy SSH keys added to a user during provisioning. For information on adding SSH keys to a user, see Adding SSH Keys to a User in *Administering Red Hat Satellite*.

Procedure

To deploy SSH keys during provisioning, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **Provisioning Templates**.
2. Create a provisioning template, or clone and edit an existing template. For more information, see Section 3.6, “Creating Provisioning Templates”.
3. In the template, click the **Template** tab.
4. In the **Template editor** field, add the **`create_users`** snippet to the **`%post`** section:

   ```
   <%= snippet('create_users') %>
   ```

5. Select the **Default** check box.
6. Click the **Association** tab.
7. From the **Application Operating Systems** list, select an operating system.
8. Click **Submit** to save the provisioning template.
9. Create a host that is associated with the provisioning template or rebuild a host using the OS associated with the modified template. For more information, see Creating a Host in the *Managing Hosts* guide.

   The SSH keys of the **Owned by** user are added automatically when the **`create_users`** snippet is executed during the provisioning process. You can set **Owned by** to an individual user or a user group. If you set **Owned by** to a user group, the SSH keys of all users in the user group are added automatically.

5.10. BUILDING A SATELLITE DISCOVERY IMAGE

Use this procedure to build a Satellite discovery image or rebuild an image if you change configuration files.

Do not use this procedure on your production Satellite or Capsule.

Prerequisites

Install the **`livecd-tools`** package:

```
# yum install livecd-tools
```

Because Anaconda installer cannot publish through HTTPS, you must enable publishing through HTTP for the repositories:

1. In the Satellite web UI, navigate to **Content** > **Products** and in the **Products** window, click the **Repositories** tab.
2. Select the **Red Hat Enterprise Linux 7** repository and for the **Publish via HTTP**, option, click the **Edit** icon, select the check box, and click **Save**.
3. Repeat the previous steps for the Satellite repository.

Procedure

To build the Satellite discovery image, complete the following steps:

1. Open the **`/usr/share/foreman-discovery-image/foreman-discovery-image.ks`** file for editing:

   ```
   # vim /usr/share/foreman-discovery-image/foreman-discovery-image.ks
   ```

2. Replace **`repo --name=rhel --baseurl=http://download/00000`** with with your own repos for RHEL and Satellite with your repository URLs. To find the URLs, navigate to **Content** > **Products** and click the **Repositories** tab and copy the URL for both repositories into the file:

   ```
   repo --name=rhel --baseurl=http://download/released/RHEL-
   ```

```
7/7.4/Server/x86_64/os/
repo --name=sat --baseurl=http://download2/nightly/Satellite
/6.4/candidate/latest-Satellite-6.4-RHEL-7/compose/Satellite
/x86_64/os/
```

3. Run the **`livecd-creator`** tool:

```
# livecd-creator --title="Discovery-Image" \
--compression-type=xz \
--cache=var/cache/build-fdi \
--config /usr/share/foreman-discovery-image/foreman-discovery-
image.ks \
--fslabel fdi \
--tmpdir /var/tmp
```

   If you change **`fdi`** in the **`--fslabel`** option, you must also change the root label on the kernel command line when loading the image. **`fdi`** or the alternative name is appended to the **`.iso`** file that is created as part of this procedure. The PXE Discovery tool uses this name when converting from **`.iso`** to PXE.

   Use **`/var/tmp`** because this process requires close to 3GB of space and **`/tmp`** might have problems if the system is low on swap space.

4. Verify that your **`fdi.iso`** file is created:

```
# ls *.iso -h
```

When you create the **`.iso`** file, you can boot the **`.iso`** file over a network or locally. Complete one of the following procedures.

To boot the iso file over a network:

1. To extract the initial ramdisk and kernel files from the **`.iso`** file over a network, enter the following command:

```
# discovery-iso-to-pxe fdi.iso
```

2. Create a directory to store your boot files:

   ```
   # mkdir /var/lib/tftpboot/boot/myimage
   ```

3. Copy the **initrd0.img** and **vmlinuz0** files to your new directory.

4. Edit the **KERNEL** and **APPEND** entries in the **/var/lib/tftpboot/pxelinux.cfg** file to add the information about your own initial ramdisk and kernel files.

To boot the iso file locally:

If you want to create a hybrid **.iso** file for booting locally, complete the following steps:

1. To convert the **.iso** file to an **.iso** hybrid file for PXE provisioning, enter the following command:

   ```
   # isohybrid --partok fdi.iso
   ```

   If you have **grub2** packages installed, you can also use the following command to install a **grub2** bootloader:

   ```
   # isohybrid --partok --uefi fdi.iso
   ```

2. To add **md5** checksum to the **.iso** file so it can pass installation media validation tests in Satellite, enter the following command:

   ```
   # implantisomd5 fdi.iso
   ```

CHAPTER 6. PROVISIONING VIRTUAL MACHINES ON A KVM

SERVER (LIBVIRT)

Kernel-based Virtual Machines (KVMs) use an open source virtualization daemon and API called **`libvirt`** running on Red Hat Enterprise Linux. Red Hat Satellite 6 can connect to the **`libvirt`** API on a KVM server, provision hosts on the hypervisor, and control certain virtualization functions.

6.1. PREREQUISITES FOR KVM PROVISIONING

The requirements for KVM provisioning include:

- Synchronized content repositories for Red Hat Enterprise Linux 7. For more information, see Synchronizing Red Hat Repositories in the *Content Management Guide*.
- A Capsule Server managing a network on the KVM server. Ensure no other DHCP services run on this network to avoid conflicts with the Capsule Server. For more information about network service configuration for Capsule Servers, see Chapter 4, *Configuring Networking*.
- An activation key for host registration. For more information, see Creating An Activation Key in the *Content Management* guide.
- A Red Hat Enterprise Linux server running KVM virtualization tools. For more information, see the Red Hat Enterprise Linux 7 Virtualization Getting Started Guide.
- An existing virtual machine image if you want to use image-based provisioning. Ensure that this image exists in a storage pool on the KVM host. The **`default`** storage pool is usually located in **`/var/lib/libvirt/images`**.

User Roles and Permissions to Provision Libvirt Compute Resources

To provision a Libvirt host in Satellite, you must have a user account with the following roles:

- **`Edit hosts`**
- **`View hosts`**

For more information, see Assigning Roles to a User in the *Administering Red Hat Satellite* guide.

You must also create a custom role with the following permissions:

- **`view_compute_resources`**
- **`destroy_compute_resources_vms`**
- **`power_compute_resources_vms`**
- **`create_compute_resources_vms`**
- **`view_compute_resources_vms`**
- **`view_locations`**
- **`view_subnets`**

For more information about creating roles, see Creating a Role. For more information about adding permissions to a role, see Adding Permissions to a Role in the *Administering Red Hat Satellite* guide.

6.2. CONFIGURING SATELLITE SERVER FOR KVM CONNECTIONS

Before adding the KVM connection, Satellite Server requires some configuration to ensure a secure connection. This means creating an SSH key pair for the user that performs the connection, which is the **`foreman`** user.

1. On Satellite Server, switch to the **`foreman`** user:

```
# su foreman -s /bin/bash
```

2. Generate the key pair:

```
$ ssh-keygen
```

3. Copy the public key to the KVM server:

```
$ ssh-copy-id root@kvm.example.com
```

4. Install the **`libvirt-client`** package:

```
# yum install libvirt-client
```

5. Use the following command to test the connection to the KVM server:

```
$ virsh -c qemu+ssh://root@kvm.example.com/system list
```

   When you add the KVM connection in Satellite Server, use the **`qemu+ssh`** protocol and the address to the server. For example, **`qemu+ssh://root@kvm.example.com/system`**.

6.3. ADDING A KVM CONNECTION TO SATELLITE SERVER

Use this procedure to add a KVM connection to Satellite Server's compute resources.

Procedure

To add a KVM connection to Satellite, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**, and in the Compute Resources window, click **Create Compute Resource**.

2. In the **Name** field, enter a name for the new compute resource.

3. From the **Provider** list, select **Libvirt**

4. In the **Description** field, enter a description for the compute resource.

5. In the **URL** field, enter the connection URL to the KVM server. For example, **`qemu+ssh://root@kvm.example.com/system`**.

6. From the **Display type** list, select either **VNC** or **Spice**.

7. Optional: To secure console access for new hosts with a randomly generated password, select the **Set a**

randomly generated password on the display connection check box. You can retrieve the password for the VNC console to access guest virtual machine console from the **`libvirtd`** host from the output of the following command:

```
# virsh edit your_VM_name
<graphics type=vnc port=-1 autoport=yes listen=0.0.0.0
passwd=your_randomly_generated_password>
```

The password is randomly generated every time the console for the virtual machine is opened, for example, with virt-manager.

8. Click **Test Connection** to ensure that Satellite Server connects to the KVM server without fault.
9. Verify that the **Locations** and **Organizations** tabs are automatically set to your current context. If you want, add additional contexts to these tabs.
10. Click **Submit** to save the KVM connection.

For CLI Users

Create the connection with the **`hammer compute-resource create`** command:

```
# hammer compute-resource create --name "My_KVM_Server" \
--provider "Libvirt" --description "KVM server at kvm.example.com"
\
--url "qemu+ssh://root@kvm.example.com/system" --locations "New
York" \
--organizations "My_Organization"
```

6.4. ADDING KVM IMAGES TO SATELLITE SERVER

If you want to use image-based provisioning to create hosts, you must add information about the image to your Satellite Server. This includes access details and the image location.

Procedure

To add KVM images on Satellite Server, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**, and in the Compute Resources window, click the name of your KVM connection.
2. Click the **Image** tab, and then click **New Image**.
3. In the **Name** field, enter a name for the image.
4. From the **Operatingsystem** list, select the image's base operating system.
5. From the **Architecture** list, select the operating system architecture.
6. In the **Username** field, enter the SSH user name for image access. This is normally the **`root`** user.
7. In the **Password** field, enter the SSH password for image access.
8. From the **User data** list, select if you want images to support user data input, such as **`cloud-init`** data.
9. In the **Image path** field, enter the full path that points to the image on the KVM server. For example, **`/var/lib/KVM/images/TestImage.qcow2`**.
10. Click **Submit** to save the image details.

For CLI Users

Create the image with the **`hammer compute-resource image create`** command. Use the **`--uuid`** field to store the full path of the image location on the KVM server.

```
# hammer compute-resource image create --name "Test KVM Image" \
--operatingsystem "RedHat version" --architecture "x86_64"
--username root \
--user-data false --uuid "/var/lib/libvirt/images/TestImage.qcow2"
\
--compute-resource "My_KVM_Server"
```

6.5. ADDING KVM DETAILS TO A COMPUTE PROFILE

We can predefine certain hardware settings for KVM-based virtual machines by adding these hardware settings to a compute profile.

Procedure

To add Red Hat Virtualization details to a compute profile, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Profiles** and in the Compute Profiles window, click the name of the KVM connection.
2. In the **CPUs** field, enter the number of CPUs to allocate to the new host.
3. In the **Memory** field, enter the amount of memory to allocate to the new host.
4. From the **Image** list, select the image to use if performing image-based provisioning.
5. From the **Network Interfaces** list, select the network parameters for the host’s network interface. You can create multiple network interfaces. However, at least one interface must point to a Capsule-managed network.
6. In the **Storage** area, enter the storage parameters for the host. You can create multiple volumes for the host.
7. Click **Submit** to save the settings to the compute profile.

For CLI Users

The compute profile CLI commands are not yet implemented in Red Hat Satellite 6.4. As an alternative, you can include the same settings directly during the host creation process.

6.6. CREATING HOSTS ON A KVM SERVER

In Satellite, you can use KVM provisioning to create hosts over a network connection and from an existing image.

If you create a host with an existing image, the new host entry triggers the KVM server to create the virtual machine, using the pre-existing image as a basis for the new volume.

If you want to create a host over a network connection, the new host must have access either to Satellite Server's integrated Capsule or an external Capsule Server on a KVM virtual network, so that the host has access to PXE provisioning services. This new host entry triggers the KVM server to create and start a virtual machine. If the virtual machine detects the defined Capsule Server through the virtual network, the virtual machine boots to PXE and begins to install the chosen operating system.

DHCP Conflicts

For network-based provisioning, if you use a virtual network on the KVM server for provisioning, select a network that does not provide DHCP assignments. This causes DHCP conflicts with Satellite Server when booting new hosts.

Procedure

To create a KVM host, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **New Host**.
2. In the **Name** field, enter the name that you want to become the provisioned system's host name.
3. Click the **Organization** and **Location** tabs to ensure that the provisioning context is automatically set to the current context.
4. From the **Host Group** list, select the host group that you want to use to populate the form.
5. From the **Deploy on** list, select the KVM connection.
6. From the **Compute Profile** list, select a profile to use to automatically populate virtual machine-based settings.
7. Click the **Interface** tab and click **Edit** on the host's interface.
8. Verify that the fields are automatically populated with values. Note in particular:
    - The **Name** from the **Host** tab becomes the **DNS name**.
    - Satellite Server automatically assigns an IP address for the new host.

9. Ensure that the **MAC address** field is blank. The KVM server assigns one to the host.

10. Verify that the **Managed**, **Primary**, and **Provision** options are automatically selected for the first interface on the host. If not, select them.

11. In the interface window, review the KVM-specific fields that are populated with settings from your compute profile. Modify these settings to suit your needs.

12. Click the **Operating System** tab, and confirm that all fields automatically contain values.

13. For network-based provisioning, ensure that the **Provisioning Method** is set to **`Network Based`**. For image-based provisioning, ensure that the **Provisioning Method** is set to **`Image Based`**

14. Click **Resolve** in **Provisioning templates** to check the new host can identify the right provisioning templates to use.

15. Click the **Virtual Machine** tab and confirm that these settings are populated with details from the host group and compute profile. Modify these settings to suit your needs.

16. Click the **Parameters** tab and ensure that a parameter exists that provides an activation key. If not, add an activation key.

17. Click **Submit** to save the host entry.

For CLI Users

Create the host with the **`hammer host create`** command and include **`--provision-method build`** to use network-based provisioning.

```
# hammer host create --name "kvm-test1" --organization
"My_Organization" \
--location "New York" --hostgroup "Base" \
--compute-resource "My_KVM_Server" --provision-method build \
--build true --enabled true --managed true \
--interface
"managed=true,primary=true,provision=true,compute_type=network,comp
ute_network=examplenetwork" \
--compute-attributes="cpus=1,memory=1073741824" \
--volume="pool_name=default,capacity=20G,format_type=qcow2"
```

Create the host with the **hammer host create** command and include **--provision-method image** to use image-based provisioning.

```
# hammer host create --name "kvm-test2" --organization
"My_Organization" \
--location "New York" --hostgroup "Base" \
--compute-resource "My_KVM_Server" --provision-method image \
--image "Test KVM Image" --enabled true --managed true \
--interface
"managed=true,primary=true,provision=true,compute_type=network,comp
ute_network=examplenetwork" \
--compute-attributes="cpus=1,memory=1073741824" \
--volume="pool_name=default,capacity=20G,format_type=qcow2"
```

For more information about additional host creation parameters for this compute resource, see Appendix B, *Additional Host Parameters for Hammer CLI*.

CHAPTER 7. PROVISIONING VIRTUAL MACHINES IN RED HAT VIRTUALIZATION

Red Hat Virtualization (version 4.0 and later) or Red Hat Enterprise Virtualization (version 3.6 and earlier) is an enterprise-grade server and desktop virtualization platform built on Red Hat Enterprise Linux.

With Red Hat Satellite 6, you can manage virtualization functions through Red Hat Virtualization’s REST API version 3. REST API version 4 is not yet supported by Satellite 6. This includes creating virtual machines and controlling their power states.

Use the following procedures to add a connection to a Red Hat Virtualization environment and provision a virtual machine.

7.1. PREREQUISITES FOR RED HAT VIRTUALIZATION PROVISIONING

The requirements for Red Hat Virtualization provisioning include:

- Synchronized content repositories for Red Hat Enterprise Linux 7. For more information, see Synchronizing Red Hat Repositories in the *Content Management Guide*.
- A Capsule Server managing a logical network on the Red Hat Virtualization environment. Ensure no other DHCP services run on this network to avoid conflicts with the Capsule Server. For more information, see Chapter 4, *Configuring Networking*.
- An existing template, other than the **blank** template, if you want to use image-based provisioning. For more information about creating templates for virtual machines, see Templates in the *Virtual Machine Management Guide*.
- An activation key for host registration. For more information, see Creating An Activation Key in the *Content Management* guide.

7.2. CREATING A RED HAT VIRTUALIZATION USER

The Red Hat Virtualization server requires an administration-like user for Satellite Server communication. For security reasons, Red Hat advises against using the **admin@internal** user for such communication. Instead, create a new Red Hat Virtualization user with the following permissions:

- System
  - Configure System
    - Login Permissions

- Network
  - Configure vNIC Profile
    - Create
    - Edit Properties
    - Delete
    - Assign vNIC Profile to VM
    - Assign vNIC Profile to Template
- Template
  - Provisioning Operations
    - Import/Export
- VM
  - Provisioning Operations
    - Create
    - Delete
    - Import/Export
    - Edit Storage
- Disk
  - Provisioning Operations
    - Create
  - Disk Profile
    - Attach Disk Profile

For more information about how to create a user and add permissions in Red Hat Virtualization, see Administering User Tasks From the Administration Portal in the *Red Hat Virtualization Administration Guide*.

7.3. ADDING A RED HAT VIRTUALIZATION CONNECTION TO SATELLITE SERVER

Use this procedure to add a Red Hat Virtualization connection to Satellite Server's compute resources.

Procedure

To add a Red Hat Virtualization connection to Satellite, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**, and in the Compute Resources window, click **Create Compute Resource**.
2. In the **Name** field, enter a name for the new compute resource.
3. From the **Provider** list, select **RHV**.
4. In the **Description** field, enter a description for the compute resource.
5. In the **URL** field, enter the connection URL for the Red Hat Virtualization Manager's API.

   For example, in RHEV 3.6 and earlier, the URL is of the following form: **`https://rhvm.example.com/api`**. In RHV 4.0 and later, the URL is of the following form: **`https://rhvm.example.com/ovirt-engine/api/v3`**.
6. Optionally, select the **Use APIv4 (experimental)** check box to evaluate the new engine API.

   Warning

   The items listed in this step are provided as Technology Previews. For further information about the scope of Technology Preview status, and associated support implications, see Technology Preview Features Support Scope.
7. In the **User** field, enter the name of a user with permissions to access Red Hat Virtualization Manager's resources.

8. In the **Password** field, enter the password of the user.
9. Click **Load Datacenters** to populate the **Datacenter** list with data centers from your Red Hat Virtualization environment.
10. From the **Datacenter** list, select a data center.
11. From the **Quota ID** list, select a quota to limit resources available to Satellite.
12. In the **X509 Certification Authorities** field, enter the certificate authority for SSL/TLS access. Alternatively, if you leave the field blank, a self-signed certificate is generated on the first API request by the server.
13. Click the **Locations** tab and select the location you want to use.
14. Click the **Organizations** tab and select the organization you want to use.
15. Click **Submit** to save the compute resource.

For CLI Users

To create a Red Hat Virtualization connection, enter the **`hammer compute-resource create`** command with **`Ovirt`** for **`--provider`** and the name of the data center you want to use for **`--datacenter`**.

```
# hammer compute-resource create \
--name "My_RHV" --provider "Ovirt" \
--description "RHV server at rhvm.example.com" \
--url "https://rhvm.example.com/api" \
--use-v4 "false" --user "Satellite_User" \
--password "My_Password" \
--locations "New York" --organizations "My_Organization" \
--datacenter "My_Datacenter"
```

Optionally, to evaluate the new engine API, change **`false`** to **`true`** for the **`--use-v4`** option.

Warning

The items listed in this step are provided as Technology Previews. For further information about the scope of Technology Preview status, and associated support implications, see Technology Preview Features Support Scope.

7.4. ADDING RED HAT VIRTUALIZATION IMAGES TO SATELLITE SERVER

Red Hat Virtualization uses templates as images for creating virtual machines. If you use image-based provisioning to create hosts, you must add Red Hat Virtualization template details to your Satellite Server. This includes access details and the template name.

Procedure

To add Red Hat Virtualization images on Satellite Server, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**, and in the Compute Resources window, click the name of your Red Hat Virtualization connection.
2. Click the **Image** tab, and then click **New Image**.
3. In the **Name** field, enter a name for the image.
4. From the **Operatingsystem** list, select the image’s base operating system.
5. From the **Architecture** list, select the operating system architecture.
6. In the **Username** field, enter the SSH user name for image access. This is normally the `root` user.
7. In the **Password** field, enter the SSH password for image access.
8. From the **Image** list, select the name of the image on Red Hat Virtualization.
9. Click **Submit** to save the image details.

For CLI Users

Create the image with the `hammer compute-resource image create` command. Use the `--uuid` option to store the template UUID on the Red Hat Virtualization server.

```
# hammer compute-resource image create --name "Test_RHV_Image" \
--operatingsystem "RedHat 7.2" --architecture "x86_64" --username
root \
--uuid "9788910c-4030-4ae0-bad7-603375dd72b1" \
--compute-resource "My_RHV"
```

7.5. ADDING RED HAT VIRTUALIZATION DETAILS TO A COMPUTE PROFILE

You can predefine certain hardware settings for virtual machines on Red Hat Virtualization. You achieve this through adding these hardware settings to a compute profile.

Procedure

To add Red Hat Virtualization details to a compute profile, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Profiles** and in the Compute Profiles window, click the name of the Red Hat Virtualization connection.
2. From the **Cluster** list, select the target host cluster in the Red Hat Virtualization environment.
3. From the **Template** list, select the RHV template to use for the **Cores** and **Memory** settings.
4. In the **Cores** field, enter the number of CPU cores to allocate to the new host.
5. In the **Memory** field, enter the amount of memory to allocate to the new host.
6. From the **Image** list, select image to use for image-based provisioning.
7. In the **Network Interfaces** area, enter the network parameters for the host’s network interface. You can create multiple network interfaces. However, at least one interface must point to a Capsule-managed network. For each network interface, enter the following details:

   a. In the **Name** field, enter the name of the network interface.

   b. From the **Network** list, select The logical network that you want to use.

8. In the **Storage** area, enter the storage parameters for the host. You can create multiple volumes for the host. For each volume, enter the following details:

   a. In the **Size (GB)** enter the size, in GB, for the new volume.

   b. From the **Storage domain** list, select the storage domain for the volume.

   c. From the **Preallocate disk**, select either thin provisioning or preallocation of the full disk.

   d. From the **Bootable** list, select whether you want a bootable or non bootable volume.

9. Click **Submit** to save the compute profile.

For CLI Users

The compute profile CLI commands are not yet implemented in Red Hat Satellite 6.4. As an alternative, you can include the same settings directly during the host creation process.

7.6. CREATING NETWORK-BASED HOSTS ON A RED HAT VIRTUALIZATION SERVER

In Satellite, you can create Red Hat Virtualization hosts over a network connection or from an existing image.

To create a host over a network, the new host must have access to either Satellite Server’s integrated Capsule or an external Capsule Server on a Red Hat Virtualization virtual network, so that the host has access to PXE provisioning services. The new host entry triggers the Red Hat Virtualization server to create the virtual machine. If the virtual machine detects the defined Capsule Server through the virtual network, the virtual machine boots to PXE and begins to install the chosen operating system.

DHCP conflicts

If you use a virtual network on the Red Hat Virtualization server for provisioning, ensure to select one that does not provide DHCP assignments. This causes DHCP conflicts with Satellite Server when booting new hosts.

When you create a host with an existing image, the new host entry triggers the Red Hat Virtualization server to create the virtual machine, using the pre-existing image as a basis for the new volume.

Procedure

To create a host for Red Hat Virtualization Server, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **New Host**.
2. In the **Name** field, enter the name that you want to become the provisioned system's host name.
3. Click the **Organization** and **Location** tabs to ensure that the provisioning context is automatically set to the current context.
4. From the **Host Group** list, select the host group that you want to use to populate the form.
5. From the **Deploy on** list, select the Red Hat Virtualization connection.
6. From the **Compute Profile** list, select a profile to use to automatically populate virtual machine-based settings.
7. Click the **Interface** tab and click **Edit** on the host's interface.
8. Verify that the fields are automatically populated with values. Note in particular:
   - The **Name** from the **Host** tab becomes the **DNS name**.
   - Satellite Server automatically assigns an IP address for the new host.
9. Ensure that the **MAC address** field is blank. The Red Hat Virtualization server assigns one to the host.
10. Verify that the **Managed**, **Primary**, and **Provision** options are automatically selected for the first interface on the host. If not, select them.
11. In the interface window, ensure that the Red Hat Virtualization-specific fields are populated with settings from the compute profile. Modify these settings to suit your needs.
12. Click the **Operating System** tab, and confirm that all fields automatically contain values.

13. For network-based provisioning, ensure that the **Provisioning Method** is set to **`Network Based`**. For image-based provisioning, ensure that the **Provisioning Method** is set to **`Image Based`**.
14. Click **Resolve** in **Provisioning templates** to check the new host can identify the right provisioning templates to use.
15. Click the **Virtual Machine** tab and confirm that these settings are populated with details from the host group and compute profile. Modify these settings to suit your needs.
16. Click the **Parameters** tab and ensure that a parameter exists that provides an activation key. If not, add an activation key.
17. Click **Submit** to save the host entry.

For CLI Users

To create a host with network-based provisioning, use the **`hammer host create`** command and include **`--provision-method build`**.

```
# hammer host create --name "rhv-test1" --organization
"My_Organization" \
--location "New York" --hostgroup "Base" \
--compute-resource "My_RHV" --provision-method build \
--build true --enabled true --managed true \
--interface
"managed=true,primary=true,provision=true,compute_name=eth0,compute
_network=satnetwork" \
--compute-
attributes="cluster=Default,cores=1,memory=1073741824,start=true" \
--volume="size_gb=20G,storage_domain=Data,bootable=true"
```

To create a host with image-based provisioning, use the **`hammer host create`** command and include **`--provision-method image`**.

```
# hammer host create --name "rhv-test2" --organization
"My_Organization" \
--location "New York" --hostgroup "Base" \
--compute-resource "My_RHV" --provision-method image \
--image "Test_RHV_Image" --enabled true --managed true \
--interface "managed=true,primary=true,provision=true,compute_name=eth0,compute_network=satnetwork" \
--compute-attributes="cluster=Default,cores=1,memory=1073741824,start=true" \
--volume="size_gb=20G,storage_domain=Data,bootable=true"
```

For more information about additional host creation parameters for this compute resource, see Appendix B, *Additional Host Parameters for Hammer CLI*.

CHAPTER 8. PROVISIONING VIRTUAL MACHINES IN VMWARE VSPHERE

VMware vSphere is an enterprise-level virtualization platform from VMware. Red Hat Satellite 6 can interact with the vSphere platform, including creating new virtual machines and controlling their power management states.

8.1. PREREQUISITES FOR VMWARE VSPHERE PROVISIONING

The requirements for VMware vSphere provisioning include:

- Synchronized content repositories for Red Hat Enterprise Linux 7. For more information, see Synchronizing Red Hat Repositories in the *Content Management Guide*.

- A Capsule Server managing a network on the vSphere environment. Ensure no other DHCP services run on this network to avoid conflicts with the Capsule Server. For more information, see Chapter 4, *Configuring Networking*.
- An existing VMware template if you aim to use image-based provisioning.
- An activation key for host registration. For more information, see Creating An Activation Key in the *Content Management* guide.

8.2. CREATING A VMWARE VSPHERE USER

The VMware vSphere server requires an administration-like user for Satellite Server communication. For security reasons, do not use the **administrator** user for such communication. Instead, create a user with the following permissions:

- All Privileges → Datastore → Allocate Space
- All Privileges → Network → Assign Network
- All Privileges → Resource → Assign virtual machine to resource pool
- All Privileges → Virtual Machine → Configuration (All)
- All Privileges → Virtual Machine → Interaction
- All Privileges → Virtual Machine → Inventory
- All Privileges → Virtual Machine → Provisioning

8.3. ADDING A VMWARE VSPHERE CONNECTION TO SATELLITE SERVER

Use this procedure to add a VMware vSphere connection in Satellite Server’s compute resources.

Ensure that the host and network-based firewalls are configured to allow Satellite to vCenter communication on

TCP port 443. Verify that Satellite is able to resolve the host name of vCenter and vCenter is able to resolve Satellite Server’s host name.

Procedure

To add a connection, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**, and in the Compute Resources window, click **Create Compute Resource**.
2. In the **Name** field, enter a name for the resource.
3. From the **Provider** list, select **VMware**.
4. In the **Description** field, enter a description for the resource.
5. In the **VCenter/Server** field, enter the IP address or host name of the vCenter server.
6. In the **User** field, enter the user name with permission to access the vCenter’s resources.
7. In the **Password** field, enter the password for the user.
8. Click **Load Datacenters** to populate the list of data centers from your VMware vSphere environment.
9. From the **Datacenter** list, select a specific data center to manage from this list.
10. In the **Fingerprint** field, ensure that this field is populated with the fingerprint from the data center.
11. From the **Display Type** list, select a console type, for example, **VNC** or **VMRC**. Note that VNC consoles are unsupported on VMware ESXi 6.5 and later.
12. Optional: In the **VNC Console Passwords** field, select the **Set a randomly generated password on the display connection** check box to secure console access for new hosts with a randomly generated password. You can retrieve the password for the VNC console to access guest virtual machine console from the `libvirtd` host from the output of the following command:

    ```
    # virsh edit your_VM_name
    <graphics type=vnc port=-1 autoport=yes listen=0.0.0.0
    passwd=your_randomly_generated_password>
    ```

The password randomly generates every time the console for the virtual machine opens, for example, with virt-manager.

13. From the **Enable Caching** list, you can select whether to enable caching of compute resources. For more information, see Section 8.7, “Caching of Compute Resources”.

14. Click the **Locations** and **Organizations** tabs and verify that the values are automatically set to your current context. You can also add additional contexts.

15. Click **Submit** to save the connection.

For CLI Users

Create the connection with the **`hammer compute-resource create`** command. Select **`Vmware`** as the **`--provider`** and set the instance UUID of the data center as the **`--uuid`**:

```
# hammer compute-resource create --name "My_vSphere" \
--provider "Vmware" \
--description "vSphere server at vsphere.example.com" \
--server "vsphere.example.com" --user "SatelliteUser" \
--password "p@55w0rd!" --locations "New York" --organizations
"My_Organization" \
--uuid 72cb9454-81cd-4231-a863-d9baf0f399f8
```

8.4. ADDING VMWARE VSPHERE IMAGES TO SATELLITE SERVER

VMware vSphere uses templates as images for creating new virtual machines. If using image-based provisioning to create new hosts, you need to add VMware template details to your Satellite Server. This includes access details and the template name.

Procedure

To add an image, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and in the Compute Resources window, click the VMware vSphere connection.

2. In the **Name** field, enter a name for the image.

3. From the **Operatingsystem** list, select the image’s base operating system.

4. From the **Architecture** list, select the operating system architecture.

5. In the **User** field, enter the SSH user name for image access. This is normally the **`root`** user.

6. In the **Password** field, enter the SSH password for image access.

7. From the **User data** list, select whether you want the images to support user data input, such as **`cloud-init`** data.

8. In the **Image** field, enter the relative path and name of the template on the vSphere environment. Do not include the data center in the relative path.

9. Click **Submit** to save the image details.

For CLI Users

Create the image with the **`hammer compute-resource image create`** command. Use the **`--uuid`** field to store the relative template path on the vSphere environment.

```
# hammer compute-resource image create --name "Test_vSphere_Image" \
--operatingsystem "RedHat 7.2" --architecture "x86_64" \
--username root --uuid "Templates/RHEL72" \
--compute-resource "My_vSphere"
```

8.5. ADDING VMWARE VSPHERE DETAILS TO A COMPUTE PROFILE

You can predefine certain hardware settings for virtual machines on VMware vSphere. You achieve this through adding these hardware settings to a compute profile.

Procedure

To add VMware vSphere details to a compute profile, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Profiles** and, in the Compute Profiles window, click the name of the compute profile, and then click the vSphere connection.
2. In the **CPUs** field, enter the number of CPUs to allocate to the new host.
3. In the **Cores per socket** field, enter the number of cores to allocate to each CPU.
4. In the **Memory** field, enter the amount of memory to allocate to the new host.
5. In the **Cluster** field, enter the name of the target host cluster on the VMware environment.
6. From the **Resource pool** list, select an available resource allocations for the host.
7. In the **Folder** field, enter the folder to organize the host.
8. From the **Guest OS** list, select the operating system you want to use in VMware vSphere.
9. From the **SCSI controller** list, select the disk access method for the host.
10. From the **Virtual H/W version** list, select the underlying VMware hardware abstraction to use for virtual machines.
11. You can select the **Memory hot add** or **CPU hot add** check boxes if you want to add more resources while the virtual machine is powered.
12. From the **Image** list, select the image to use if performing image-based provisioning.
13. From the **Network Interfaces** list, select the network parameters for the host’s network interface. You can create multiple network interfaces. However, at least one interface must point to a Capsule-managed network.
14. Select the **Eager zero** check box if you want to use eager zero thick provisioning. If unchecked, the disk uses lazy zero thick provisioning.
15. Click **Submit** to save the compute profile.

For CLI Users

The compute profile CLI commands are not yet implemented in Red Hat Satellite 6.4. As an alternative, you can include the same settings directly during the host creation process.

8.6. CREATING HOSTS ON A VMWARE VSPHERE SERVER

The VMware vSphere provisioning process provides the option to create hosts over a network connection or using an existing image.

For network-based provisioning, you must create a host to access either Satellite Server's integrated Capsule or an external Capsule Server on a VMware vSphere virtual network, so that the host has access to PXE provisioning services. The new host entry triggers the VMware vSphere server to create the virtual machine. If the virtual machine detects the defined Capsule Server through the virtual network, the virtual machine boots to PXE and begins to install the chosen operating system.

DHCP Conflicts

If you use a virtual network on the VMware vSphere server for provisioning, ensure that you select a virtual network that does not provide DHCP assignments. This causes DHCP conflicts with Satellite Server when booting new hosts.

For image-based provisioning, use the pre-existing image as a basis for the new volume.

Procedure

To create a host for a VMware vSphere server, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **New host**.
2. In the **Name** field, enter the name that you want to become the provisioned system's host name.
3. Click the **Organization** and **Location** tabs to ensure that the provisioning context is automatically set to the current context.
4. From the **Host Group** list, select the host group that you want to use to populate the form.

5. From the **Deploy on** list, select the VMware vSphere connection.
6. From the **Compute Profile** list, select a profile to use to automatically populate virtual machine-based settings.
7. Click the **Interface** tab and click **Edit** on the host's interface.
8. Verify that the fields are automatically populated with values. Note in particular:
   - The **Name** from the **Host** tab becomes the **DNS name**.
   - The Satellite Server automatically assigns an IP address for the new host.
9. Ensure that the **MAC address** field is blank. The VMware vSphere server assigns one to the host.
10. Verify that the **Managed**, **Primary**, and **Provision** options are automatically selected for the first interface on the host. If not, select them.
11. In the interface window, review the VMware vSphere-specific fields that are populated with settings from our compute profile. Modify these settings to suit your needs.
12. Click the **Operating System** tab, and confirm that all fields automatically contain values.
13. For network-based provisioning, ensure that the **Provisioning Method** is set to `Network Based`. For image-based provisioning, ensure that the **Provisioning Method** is set to `Image Based`
14. Click **Resolve** in **Provisioning templates** to check the new host can identify the right provisioning templates to use.
15. Click the **Virtual Machine** tab and confirm that these settings are populated with details from the host group and compute profile. Modify these settings to suit your needs.
16. Click the **Parameters** tab and ensure that a parameter exists that provides an activation key. If not, add an activation key.
17. Click **Submit** to save the host entry.

For CLI Users

Create the host from a network with the `hammer host create` command and include `--provision-method build` to use network-based provisioning.

```
# hammer host create --name "vmware-test1" --organization
"_My_Organization_" \
--location "New York" --hostgroup "Base" \
--compute-resource "_My_vSphere_" --provision-method build \
--build true --enabled true --managed true \
--interface
"managed=true,primary=true,provision=true,compute_type=VirtualE1000
,compute_network=mynetwork" \
--compute-
attributes="cpus=1,corespersocket=2,memory_mb=1024,cluster=MyCluste
r,path=MyVMs,start=true" \
--volume="size_gb=20G,datastore=Data,name=myharddisk,thin=true"
```

Note

See Appendix B, *Additional Host Parameters for Hammer CLI* for more information on additional host creation parameters for this compute resource.

For CLI Users

Create the host from an image with the **`hammer host create`** command and include **`--provision-method image`** to use image-based provisioning.

```
# hammer host create --name "vmware-test2" --organization
"My_Organization" \
--location "New York" --hostgroup "Base" \
--compute-resource "My_RHEV" --provision-method image \
--image "Test RHEV Image" --enabled true --managed true \
--interface
"managed=true,primary=true,provision=true,compute_type=VirtualE1000
,compute_network=mynetwork" \
--compute-
attributes="cpus=1,corespersocket=2,memory_mb=1024,cluster=MyCluste
```

```
r,path=MyVMs,start=true" \
--volume="size_gb=20G,datastore=Data,name=myharddisk,thin=true"
```

For more information about additional host creation parameters for this compute resource, see Appendix B, *Additional Host Parameters for Hammer CLI*.

8.7. CACHING OF COMPUTE RESOURCES

Caching of compute resources speeds up rendering of VMware information.

8.7.1. Enabling Caching of Compute Resources

To enable or disable caching of compute resources:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**.
2. Click the **Edit** button to the right of the VMware server you want to update.
3. Select the **Enable caching** check box.

8.7.2. Refreshing the Compute Resources Cache

To refresh the cache of compute resources to update compute resources information:

Procedure

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources**.
2. Select a VMware server you want to refresh the compute resources cache for and click the **Refresh Cache** button.

For CLI Users

Use this API call to refresh the compute resources cache:

```
# curl -H "Accept:application/json,version=2" \
-H "Content-Type:application/json" -X PUT \
-u username:password -k \
https://satellite.example.com/api/compute_resources
/compute_resource_id/refresh_cache
```

Use the **`hammer compute-resource list`** command to determine the ID of the VMware server you want to refresh the compute resources cache for.

CHAPTER 9. PROVISIONING CLOUD INSTANCES IN RED HAT OPENSTACK PLATFORM

Red Hat OpenStack Platform provides the foundation to build a private or public Infrastructure-as-a-Service (IaaS) cloud on Red Hat Enterprise Linux. It offers a massively scalable, fault-tolerant platform for the development of cloud-enabled workloads. In Red Hat Satellite 6, you can interact with Red Hat OpenStack Platforms REST API to create new cloud instances and control their power management states.

9.1. PREREQUISITES FOR RED HAT OPENSTACK PLATFORM PROVISIONING:

Requirements for Red Hat OpenStack Platform Provisioning include:

- Synchronized content repositories for Red Hat Enterprise Linux 7. For more information, see Synchronizing Red Hat Repositories in the *Content Management Guide*.

- A Capsule Server managing a network in your OpenStack environment. For more information, see Chapter 4, *Configuring Networking*.
- An image added to OpenStack Image Storage (glance) service for image-based provisioning. For more information, see the Red Hat OpenStack Platform Instances and Images Guide.
- An activation key for host registration. For more information, see Creating An Activation Key in the *Content Management* guide.

9.2. ADDING A RED HAT OPENSTACK PLATFORM CONNECTION TO THE SATELLITE SERVER

Use this procedure to add the Red Hat OpenStack Platform connection in the Satellite Server's compute resources.

Procedure

To add a compute resource, use the following procedure:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and in the Compute Resources window, click **Create Compute Resource**.
2. In the **Name** field, enter a name to identify the compute resource for future use.
3. From the **Provider** list, select **RHEL OpenStack Platform**.
4. In the **Description** field, enter a description for the resource.
5. In the **URL** field, enter a URL to point to the OpenStack Authentication keystone service's API at the `tokens` resource. Use the following format: `http://openstack.example.com:5000/v2.0/tokens`.
6. In the **User** and **Password** fields, enter the authentication user and password for Satellite to access the environment.
7. In the **Domain** field, enter the domain for V3 authentication.

8. From the **Tenant** list, select the tenant or project for Satellite Server to manage.
9. To use external networks as primary networks for hosts, select the **Allow external network as main network** check box.
10. Click the **Locations** and **Organizations** tabs and verify that the location and organization that you want to use are set to your current context. Add any additional contexts that you want to these tabs.
11. Click **Submit** to save the Red Hat OpenStack Platform connection.

For CLI Users

Create the connection with the **`hammer compute-resource create`** command:

```
# hammer compute-resource create --name "My_OpenStack" \
--provider "OpenStack" \
--description "My OpenStack environment at openstack.example.com" \
--url "http://openstack.example.com:5000/v2.0/tokens" --user
"My_Username" \
--password "My_Password" --tenant "openstack" --locations "New
York" \
--organizations "My_Organization"
```

9.3. ADDING RED HAT OPENSTACK PLATFORM IMAGES TO THE SATELLITE SERVER

Red Hat OpenStack Platform uses image-based provisioning to create new hosts. You must add image details to your Satellite Server. This includes access details and image location.

Procedure

To add Red Hat OpenStack Platform images, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and in the Computer Resources

window, click the name of a Red Hat OpenStack Platform connection.

2. In the **Name** field, enter a name that describes the image.
3. From the **Operatingsystem** list, select the image’s base operating system.
4. From the **Architecture** list, select the operating system architecture.
5. In the **User** field, enter the SSH user name for image access. This is normally the **`root`** user.
6. In the **Password** field, enter the SSH password for image access.
7. From the **Image** list, select the image in OpenStack Image Storage.
8. From the **User data** list, select whether you want to set if images support user data input, such as **`cloud-init`** data.
9. Click **Submit** to save the image details.

For CLI Users

Create the image with the **`hammer compute-resource image create`** command. Use the **`--uuid`** field to store the full path of the image location on the Red Hat OpenStack Platform server.

```
# hammer compute-resource image create --name "Test OpenStack
Image" \
--operatingsystem "RedHat 7.2" --architecture "x86_64" \
--user root --user-data true \
--compute-resource "My_OpenStack_Platform"
```

9.4. ADDING RED HAT OPENSTACK PLATFORM DETAILS TO A COMPUTE PROFILE

In Satellite, you can define certain hardware settings for instances on Red Hat OpenStack Platform. You can add these hardware settings to a compute profile.

Procedure

To add Red Hat OpenStack Platform details to a compute profile, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Profiles** and in the Compute Profiles window, click the name of the profile you want to use.
2. From the **Flavor** list, select the hardware profile on OpenStack Platform to use for the host.
3. From the **Availability zone** list, selec the target cluster to use within the OpenStack Platform environment.
4. From the **Image** list, select the image to use for image-based provisioning.
5. From the **Tenant** list, select the tenant or project for the OpenStack Platform instance.
6. From the **Security Group** list, select the cloud-based access rules for ports and IP addresses.
7. From the **Internal network**, select the private networks for the host to join.
8. From the **Floating IP network**, select the external networks for the host to join and assign a floating IP address.
9. From the **Boot from volume**, select whether a volume is created from the image. If not selected, the instance boots the image directly.
10. In the **New boot volume size (GB)** field, enter the size, in GB, of the new boot volume.
11. Click **Submit** to save the compute profile.

For CLI Users

The compute profile CLI commands are not yet implemented in Red Hat Satellite 6.4. As an alternative, you can include the same settings directly during the host creation process.

9.5. CREATING IMAGE-BASED HOSTS ON RED HAT OPENSTACK PLATFORM

In Satellite, you can provision Red Hat OpenStack Platform hosts from existing images on the Red Hat OpenStack

Platform server.

Procedure

To provision a host, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts** > **New Host**.
2. In the **Name** field, enter the name that you want to assign to the provisioned system's host.
3. From the **Host Group** list, you can select a host group to populate the host fields.
4. From the **Deploy on** lits, select the OpenStack Platform connection.
5. From the **Compute profile** list, select a profile to use to automatically populate cloud instance-based settings.
6. Click the **Interface** tab, and click **Edit** on the host's interface. Verify that the **Name** from the **Host** tab becomes the **DNS name**, and that the Satellite Server automatically assigns an IP address for the new host.
7. Ensure that the **MAC address** field is blank. The Red Hat OpenStack Platform server assigns a MAC address to the host.
8. Verify that Satellite Server automatically selects the **Managed**, **Primary**, and **Provision** options for the first interface on the host. If not, select them.
9. Click the **Operating System** tab, and confirm that each aspect of the operating system is populated.
10. If you want to change the image that populates automatically from your compute profile, from the **Images** list, select a different image to base the new host's root volume.
11. Click **Resolve** in **Provisioning Templates** to verify that the new host can identify the right provisioning templates to use.
12. Click the **Virtual Machine** tab, and verify that the settings are populated with details from the host groups and compute profile. Modify these settings to suit your needs.
13. Click the **Parameters** tab and ensure that a parameter exists that provides an activation key. If not, add an activation key.

14. Click **Submit** to save the changes.

This new host entry triggers the Red Hat OpenStack Platform server to create the instance, using the pre-existing image as a basis for the new volume.

For CLI Users

Create the host with the **`hammer host create`** command and include the **`--provision-method image`** option to use image-based provisioning.

```
# hammer host create --name "openstack-test1" --organization
"My_Organization" \
--location "New York" --hostgroup "Example_Hostgroup" \
--compute-resource "My_OpenStack_Platform" --provision-method image
\
--image "Test OpenStack Image" --enabled true --managed true \
--interface "managed=true,primary=true,provision=true" \
--compute-
attributes="flavor_ref=m1.small,tenant_id=openstack,security_groups
=default,network=mynetwork"
```

For more information about additional host creation parameters for this compute resource, see Appendix B, *Additional Host Parameters for Hammer CLI*.

CHAPTER 10. PROVISIONING CLOUD INSTANCES IN AMAZON EC2

Amazon Elastic Compute Cloud (Amazon EC2) is a web service that provides public cloud compute resources.

Using Red Hat Satellite 6, you can interact with Amazon EC2's public API to create cloud instances and control their power management states. Use the procedures in this chapter to add a connection to an Amazon EC2 account and provision a cloud instance.

10.1. PREREQUISITES FOR AMAZON EC2 PROVISIONING

The requirements for Amazon EC2 provisioning include:

- Synchronized content repositories for Red Hat Enterprise Linux 7. For more information, see Synchronizing Red Hat Repositories in the *Content Management Guide*.
- A Capsule Server managing a network in your EC2 environment. Use a Virtual Private Cloud (VPC) to ensure a secure network between the hosts and the Capsule Server.
- An Amazon Machine Image (AMI) for image-based provisioning.
- An activation key for host registration. For more information, see Creating An Activation Key in the *Content Management* guide.

10.2. ADDING AN AMAZON EC2 CONNECTION TO THE SATELLITE SERVER

Use this procedure to add the Amazon EC2 connection in the Satellite Server's compute resources.

Time Settings and Amazon Web Services

Amazon Web Services uses time settings as part of the authentication process. Ensure that Satellite Server's time is correctly synchronized. Ensure that an NTP service, such as **`ntpd`** or **`chronyd`**, is running properly on the Satellite Server. Failure to provide the correct time to Amazon Web Services can lead to authentication failures. For more information about synchronizing time in Satellite, see Synchronizing Time in *Installing Satellite Server from a Connected Network*.

Procedure

To add an Amazon EC2 connection, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and in the Compute Resources window, click **Create Compute Resource**.
2. In the **Name** field, enter a name to identify the Amazon EC2 compute resource.
3. From the **Provider** list, select **EC2**.
4. In the **Description** field, enter information that helps distinguish the resource for future use.
5. From the **HTTP proxy** list, select a HTTP proxy to connect to external API services. You must add HTTP proxies to Satellite before you can select a proxy from this list. For more information, see Configuring Satellite Server with HTTP Proxy in *Installing Satellite Server from a Connected Network*.
6. In the **Access Key** and **Secret Key** fields, enter the access keys for your Amazon EC2 account. For more information, see Managing Access Keys for your AWS Account on the Amazon documentation website.
7. Click the **Load Regions** button to populate the **Regions** list.
8. From the **Region** list, select the Amazon EC2 region or data center to use.
9. Click the **Locations** tab and ensure that the location you want to use is selected, or add a different location.
10. Click the **Organizations** tab and ensure that the organization you want to use is selected, or add a different organization.
11. Click **Submit** to save the Amazon EC2 connection.

For CLI Users

Create the connection with the **`hammer compute-resource create`** command. Use **`--user`** and **`--password`** options to add the access key and secret key respectively.

```
# hammer compute-resource create --name "My_EC2" --provider "EC2" \
--description "Amazon EC2 Public Cloud` --user "user_name" \
--password "secret_key" --region "us-east-1" --locations "New York"
\
--organizations "My_Organization"
```

10.3. ADDING AMAZON EC2 IMAGES TO SATELLITE SERVER

Amazon EC2 uses image-based provisioning to create hosts. You must add image details to your Satellite Server. This includes access details and image location.

Procedure

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and select an Amazon EC2 connection.
2. Click the **Images** tab, and then click **New Image**.
3. In the **Name** field, enter a name to identify the image for future use.
4. From the **Operatingsystem** list, select the operating system that corresponds with the image you want to add.
5. From the **Architecture** list, select the operating system's architecture.
6. In the **Username** field, enter the SSH user name for image access. This is normally the `root` user.
7. In the **Password** field, enter the SSH password for image access.
8. In the **Image ID** field, enter the Amazon Machine Image (AMI) ID for the image. This is usually in the following format: `ami-xxxxxxxx`.
9. From the **User data** list, select whether the images support user data input, such as `cloud-init` data. If you enable user data, the Finish scripts are automatically disabled. This also applies in reverse: if you enable the Finish scripts, this disables user data.
10. From the **IAM role** list, select the Amazon security role used for creating the image.
11. Click **Submit** to save the image details.

For CLI Users

Create the image with the **`hammer compute-resource image create`** command. Use the **`--uuid`** field to store the full path of the image location on the Amazon EC2 server.

```
# hammer compute-resource image create --name "Test Amazon EC2
Image" \
--operatingsystem "RedHat 7.2" --architecture "x86_64" --username
root \
--user-data true --uuid "ami-my_ami_id" --compute-resource "My_EC2"
```

10.4. ADDING AMAZON EC2 DETAILS TO A COMPUTE PROFILE

You can add hardware settings for instances on Amazon EC2 to a compute profile.

Procedure

To add hardware settings, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Profiles** and click the name of your profile, then click an EC2 connection.
2. From the **Flavor** list, select the hardware profile on EC2 to use for the host.
3. From the **Image** list, select the image to use for image-based provisioning.
4. From the **Availability zone** list, select the target cluster to use within the chosen EC2 region.
5. From the **Subnet** list, add the subnet for the EC2 instance. If you have a VPC for provisioning new hosts, use its subnet.
6. From the **Security Groups** list, select the cloud-based access rules for ports and IP addresses to apply to the host.
7. From the **Managed IP** list, select either a **`Public`** IP or a **`Private`** IP.

8. Click **Submit** to save the compute profile.

For CLI Users

The compute profile CLI commands are not yet implemented in Red Hat Satellite 6.4. As an alternative, you can include the same settings directly during the host creation process.

10.5. CREATING IMAGE-BASED HOSTS ON AMAZON EC2

The Amazon EC2 provisioning process creates hosts from existing images on the Amazon EC2 server.

Procedure

1. In the Satellite web UI, navigate to **Hosts** > **New Host**.
2. In the **Name** field, enter a name for the host.
3. From the **Host Group** list, you can select a host group to populate most of the new host's fields.
4. From the **Deploy on** list, select the EC2 connection.
5. From the **Compute Profile** list, select a profile to use to automatically populate virtual machine-based settings.
6. Click the **Interface** tab, and then click **Edit** on the host's interface, and verify that the fields are populated with values. Leave the **Mac Address** field blank. The Satellite Server automatically selects and IP address and the **Managed**, **Primary**, and **Provision** options for the first interface on the host.
7. Click the **Operating System** tab and confirm that all fields are populated with values.
8. Click the **Virtual Machine** tab and confirm that all fields are populated with values.
9. Click the **Parameters** tab, and ensure that a parameter exists that provides an activation key. If not, add an activation key.
10. Click **Submit** to save your changes.

This new host entry triggers the Amazon EC2 server to create the instance, using the pre-existing image as a basis for the new volume.

For CLI Users

Create the host with the **`hammer host create`** command and include **`--provision-method image`** to use image-based provisioning.

```
# hammer host create --name "ec2-test1" --organization
"My_Organization" \
--location "New York" --hostgroup "Base" \
--compute-resource "My_EC2" --provision-method image \
--image "Test Amazon EC2 Image" --enabled true --managed true \
--interface
"managed=true,primary=true,provision=true,subnet_id=EC2" \
--compute-
attributes="flavor_id=m1.small,image_id=TestImage,availability_zone
s=us-east-1a,security_group_ids=Default,managed_ip=Public"
```

For more information about host creation parameters for this compute resource, see Appendix B, *Additional Host Parameters for Hammer CLI*.

10.6. CONNECTING TO AN AMAZON EC2 INSTANCE USING SSH

You can connect remotely to an Amazon EC2 instance from Satellite Server using SSH. However, to connect to any Amazon Web Services EC2 instance that you provision through Red Hat Satellite, you must first access the private key that is associated with the compute resource in the Foreman database, and use this key for authentication.

To locate the private key and connect to an Amazon EC2 server using SSH, complete the following steps:

1. To locate the compute resource list, on your Satellite Server base system, enter the following command, and note the ID of the compute resource that you want to use:

```
# hammer compute-resource list
```

2. Switch user to the **postgres** user:

```
# su - postgres
```

3. Initiate the **postgres** shell:

```
$ psql
```

4. Connect to the Foreman database as the user **postgres**:

```
# postgres=# \c foreman
```

5. Select the secret from **key_pairs** where **compute_resource_id = 3**:

```
# select secret from key_pairs where compute_resource_id = 3;
secret
```

6. Copy the key from after **-----BEGIN RSA PRIVATE KEY-----** until **-----END RSA PRIVATE KEY-----**.

7. Create a **.pem** file and paste your key into the file:

```
# vim Keyname.pem
```

8. Ensure that you restrict access to the **.pem** file:

```
# chmod 600 Keyname.pem
```

9. To connect to the Amazon EC2 instance, enter the following command:

```
ssh -i Keyname.pem   ec2-user@example.aws.com
```

10.7. CONFIGURING A FINISH TEMPLATE FOR AN AMAZON WEB SERVICE EC2 ENVIRONMENT

You can use Red Hat Satellite finish templates during the provisioning of Red Hat Enterprise Linux instances in an Amazon EC2 environment.

To configure a finish template for Amazon EC2, complete the following steps:

1. In the Red Hat Satellite 6 web UI, navigate to **Hosts** > **Provisioning Templates**.
2. In the **Provisioning Templates** page, enter `Kickstart default finish` into the search field and click **Search**.
3. On the **Kickstart default finish** template, select **Clone**.
4. In the **Name** field, enter a unique name for the template.
5. In the template, prefix each command that requires root privileges with `sudo`, except for `subscription-manager register` and `yum` commands, or add the following line to run the entire template as the sudo user:

   ```
   sudo -s << EOS
   _Template_ _Body_
   EOS
   ```

6. Click the **Association** tab, and associate the template with a Red Hat Enterprise Linux operating system that you want to use.
7. Click the **Locations** tab, and add the the location where the host resides.
8. Click the **Organizations** tab, and add the organization that the host belongs to.
9. Make any additional customizations or changes that you require, then click **Submit** to save your template.
10. Navigate to **Hosts** > **Operating systems** and select the operating system that you want for your host.
11. Click the **Templates** tab, and from the **Finish Template** list, select your finish template.

12. Navigate to **Hosts** > **Create Host** and enter the information about the host that you want to create.

13. Click the **Parameters** tab and navigate to **Host parameters**.

14. In **Host parameters**, click the **Add Parameter** button three times to add three new parameter fields. Add the following three parameters:

    a. In the **Name** field, enter **`remote_execution_ssh_keys`**. In the corresponding **Value** field, enter the output of **`cat /usr/share/foreman/.ssh/id_rsa_foreman_proxy.pub`**.

    b. In the **Name** field, enter **`remote_execution_ssh_user`**. In the corresponding **Value** field, enter **`ec2-user`**.

    c. In the **Name** field, enter **`activation_keys`**. In the corresponding **Value** field, enter your activation key.

15. Click **Submit** to save the changes.

10.8. MORE INFORMATION ABOUT AMAZON WEB SERVICES AND SATELLITE

For information about how to locate Red Hat Gold Images on Amazon Web Services EC2, see How to Locate Red Hat Cloud Access Gold Images on AWS EC2.

For information about how to install and use the Amazon Web Service Client on Linux, see Install the AWS Command Line Interface on Linux in the Amazon Web Services documentation.

For information about importing and exporting virtual machines in Amazon Web Services, see VM Import/Export in the Amazon Web Services documentation.

CHAPTER 11. PROVISIONING

CONTAINERS

With Red Hat Satellite 6, you can create an on-premise registry, import images from various sources and distribute them to containers using Content Views. Satellite Server supports creating one or more Docker compute resources that act as servers for running containers.

You can import an image, start a container based on this image, monitor the container’s activity, and commit its state to a new image layer that can be further propagated.

For information about containers, see the Getting Started with Containers guide for *Red Hat Enterprise Linux Atomic Host 7*.

Use this chapter to import container images and use these images to create containers.

11.1. PREREQUISITES FOR CONTAINER PROVISIONING

Before you can provision a container using Red Hat Enterprise Linux Atomic Host, you must have a source for images, such as a container registry.

Red Hat Satellite 6 uses three sources of container images:

- Synchronized Docker-formatted container images that are a part of the Satellite Server’s application life cycle.
- Public images from Docker Hub.
- Other External registries, including Red Hat’s container image registry. This is explored in Section 11.3, “Adding External Registries to the Satellite Server”.

11.2. IMPORTING CONTAINER IMAGES

You can import container images from the Red Hat Container Catalog or from other image registries that you connect to Satellite.

To import a container image, follow these steps:

1. Create a custom product for Red Hat Container Catalog.
2. Add a repository that links to the container catalog that you want to use. For example, Red Hat Container Catalog registry (http://registry.access.redhat.com/) or a third-party registry in Satellite.
3. Synchronize with the registry's repository.

To create a custom product for the Red Hat container Catalog and create an image repository, complete the following steps:

1. In the Satellite web UI, navigate to **Content** > **Products** and click **New Product**.
2. In the **Name** field, enter a name for the product.
3. Enter and select any additional details for the product.
4. Click **Save** to create the product.
5. Click **Create Repository**.
6. In the **Name** field, enter a name that describes the repository.
7. In the **Label** field, enter an internal ID for the repository.
8. From the **Type** list, select **`docker`**.
9. In the **URL** field, enter the URL of the registry to use as a source: for example, **`http://registry.access.redhat.com/`** or the URL of another source that you want to use, such as **`https://registry.hub.docker.com`**.
10. In the **Upstream Repository Name** field, enter the name of the repository.
11. Click **Save** to create the repository and view it in Satellite's repository list.
12. Select the new repository and then click **Sync Now** to start the synchronization process.

To view the progress of the synchronization in the Web UI. Navigate to **Content** > **Sync Status** and expand the repository tree.

When the synchronization completes, you can click **Manage Docker Manifests** to list the available manifests. From the list, you can also remove any manifests that you do not require.

For CLI Users

1. Create the custom `Red Hat Container Catalog` product:

```
# hammer product create \
--name "Red Hat Container Catalog" \
--sync-plan "Example Plan" \
--description "Red Hat Container Catalog content" \
--organization "My_Organization"
```

2. Create the repository for the container images:

```
# hammer repository create \
--name "RHEL7" \
--content-type "docker" \
--url "http://registry.access.redhat.com/" \
--docker-upstream-name "rhel7" \
--product "Red Hat Container Catalog" \
--organization "My_Organization"
```

3. Synchronize the repository:

```
# hammer repository synchronize \
--name "RHEL7" \
--product "Red Hat Container Catalog" \
--organization "My_Organization"
```

11.3. ADDING EXTERNAL REGISTRIES TO THE SATELLITE SERVER

If you want to create a container from an image in an external registry, you must first add the registry to Satellite.

To add an external container registry, complete the following steps:

1. In the Satellite web UI, navigate to **Containers** > **Registries** and click **Create Registry**.
2. In the **Name** field, enter a name for the registry.
3. In the **URL** field, enter the location of the registry. For example: **`https://registry.access.redhat.com`**.
4. Optional: In the **Description** field, enter a description for your registry entry.
5. In the **Username** field, enter the user name that corresponds with your user account on the registry.
6. In the **Password** field, enter the password for your user account on your registry.
7. Select the **Locations** tab, and select a location.
8. Select the **Organizations** tab, and select an organization.
9. Click **Submit** to save the external registry.

For CLI Users

Create the registry with the **`hammer docker registry create`** command:

```
# hammer docker registry create --name "Red Hat" \
--url "https://registry.access.redhat.com" \
--description "Red Hat Container Image Registry" \
--organization "Default_Organization" \
--location "Default_Location"
```

11.4. MANAGING CONTAINER IMAGES IN SATELLITE

To manage container images with Content Views, complete the following steps:

1. In the Satellite web UI, navigate to **Content** > **Content Views** and click **Create New View**.
2. In the **Name** field, enter **`Containers`**. This automatically populates the **Label** field.
3. In the **Description** field, enter a description. For example, **`Container image for Red Hat Enterprise Linux 7`**.
4. If you want to use a Composite Content View to hold other Content Views, select the **Composite View** check box.
5. Optional: If you select **Composite Content View**, you can select whether you want to Auto publish a composite view when a new version of a component Content View is created.
6. Click **Save** to create the Content View.
7. Navigate to the **Docker Content** subtab, then click **Add**.
8. Select the container repository for a Red Hat Enterprise Linux 7 Server image.
9. Click **Add Repository**.
10. Navigate to **Versions** and click **Publish New Version**.

You can enter a **Description** for the version; meaningful descriptions can help in logging new content versions.

Satellite Server creates the new version of the view and publishes it to the Library environment.

You can also click **Promote** to promote this Content View across environments in the application life cycle.

For CLI Users

1. To obtain a list of repository IDs:

```
# hammer repository list --organization "My_Organization"
```

2. Create the Content View and add the repository:

```
# hammer content-view create \
--name "Containers" \
--description "Container image for Red Hat Enterprise Linux 7"
```

```
\
--repository-ids 8 \
--organization "My_Organization"
```

3. Publish the view:

```
# hammer content-view publish \
--name "Containers" \
--description "Initial Content View for our container image" \
--organization "My_Organization"
```

11.5. CONFIGURING THE RED HAT ENTERPRISE LINUX ATOMIC HOST

Configure the Atomic Host before connecting to Satellite. This includes exposing the Red Hat API for Docker-formatted containers to the Satellite Server. For information about containers, see the Getting Started with Containers guide for *Red Hat Enterprise Linux Atomic Host 7*.

1. Log on to the Atomic Host and edit the **/etc/sysconfig/docker** file:

```
$ vi /etc/sysconfig/docker
```

2. Find the **OPTIONS** parameter and modify it to expose the API:

```
OPTIONS=--selinux-enabled -H unix:///var/run/docker.sock -H
tcp://0.0.0.0:2375
```

Important

Use either TCP port 2375 or 2376 for the connection. Satellite Server contains certain SELinux rules to permit access to these ports. Using an alternative port results in authentication failure.

3. Open port 2375 for the firewall:

```
# firewall-cmd --add-port=2375/tcp
# firewall-cmd --add-port=2375/tcp --permanent
```

4. Import the Satellite Server certificate:

```
$ curl http://satellite.example.com/pub/katello-server-ca.crt \
-o /etc/pki/ca-trust/source/anchors/katello-server-ca.crt
$ update-ca-trust
```

5. Restart the `docker` service:

```
# systemctl restart docker
```

11.6. ADDING AN ATOMIC HOST CONNECTION TO THE SATELLITE SERVER

To add an Red Hat Enterprise Linux Atomic connection in the Satellite Server's compute resources, complete the following steps:

1. In the Satellite web UI, navigate to **Infrastructure** > **Compute Resources** and click **Create Compute Resource**.
2. In the **Name** field, enter a name for the resource.
3. From the **Provider** list, select **Docker**.
4. Optional: In the **Description** field, you can add a description for the resource.
5. Select the **Locations** tab, and select the location that you want to use.
6. Select the **Organizations** tab, and select the organization that you want to use.
7. Click **Submit** to save.

For CLI Users

Create the connection with the **`hammer compute-resource create`** command:

```
# hammer compute-resource create --provider docker \
--name "Atomic" --url "http://atomic.example.com:2375" \
--organizations 'Default Organization' --locations 'Default
Location'
```

11.7. CREATING A CONTAINER

Use this procedure to create a container in Satellite. The Satellite web UI contains a wizard that guides you through the creation process. You can select a container image from a Content View, a Docker Hub, or an external registry that you add to Satellite.

1. In the Satellite web UI, navigate to **Containers** > **Create Container**.
2. From the **Deploy on** list, select the compute resource that you want to use.
3. Select the **Locations** tab, and select a location.
4. Select the **Organizations** tab, and select an organization.
5. Click **Next step**.

To create a container, you have three options:

- Create from a container image in Content View
- Create from an container image in the Docker Hub
- Create from an container image in an external registry.

Creating a Container from a Content View

1. From the **Lifecycle Environment** list, select the lifecycle environment that you want to use.

2. From the **Content View** list, select the container image that you want to use.
3. From the **Registry** list, select the registry that you want to use.
4. From the **Tag** list, select the container image tag that you want to use.
5. From the **Capsule** list, select the Capsule that you want to use.
6. Click **Next Step**.

Creating a Container from Docker Hub

1. Select the **Docker Hub** tab, and in the **Search** field, enter the Docker container that you want to use.
2. In the **Tag** field, enter the name of the container image tag that you want to use, and click **Search for images**.
3. Select the container image that you want to use, and click **Next Step**.

Creating a Container from an External Registry

1. From the **Registry** list, select a registry that you want to use.
2. In the **Search** field, enter the name of the container image that you want to use.
3. In the **Tag** field, enter the tag that is associated with the container image you want to use.
4. Click **Search for images** and select the image that you want to use, and click **Next step**.

Finishing the Container Creation Process

1. In the **Name** field, enter a name for the new container.
2. In the **Command** field, enter a command that you want to run in the container.
3. In the **Entry point** field, enter a command that you want the container to execute automatically when the container starts. The default entrypoint is `/bin/sh -c`.
4. Select the **Compute options** tab.
5. In the **CPU Sets** field, assign CPUs to the container. For example, 0-2,16 represents CPUs 0, 1, 2, and 16.

6. In the **CPU share** field, assign the CPU share for the container. This sets the share of CPU time available to containerized tasks.
7. In the **Memory** field, enter the memory size that you want to allocate to the container.
8. Click **Next Step**.
9. In the **Environment variables** field, define a set of environmental variables. For example, `LANG=en_US.UTF-8`.
10. In the **Exposed Ports** field, enter the number of ports that you want to open in the container. For example, you can open SSH communication to the container on port 22.
11. In the **DNS** field, enter the DNS server for the container.
12. Select the **Run** check box to start the container automatically after it is created.
13. Click **Submit** to create a container.

For CLI Users

The following are three examples of the `hammer docker container create` command.

- To create a container from a Content View:

```
# hammer docker container create --compute-resource "Atomic" \
--repository-name "rhel7" --tag "latest" --name "docker-test1" \
--command "bash" --organizations "My_Organization" --locations
"New York"
```

- To provision from the Docker Hub:

```
# hammer docker container create --compute-resource "Atomic" \
--repository-name "docker.io/redhat" --tag latest \
--name "docker-test2" --command bash --organizations
"My_Organization" \
--locations "New York"
```

- To provision from an external registry:

```
# hammer docker container create --compute-resource "Atomic" \
--registry-id 1 --repository-name "rhel" --tag latest \
--name "docker-test3 --command bash --organizations
"My_Organization" \
--locations "New York"
```

11.8. STARTING, COMMITTING, AND REMOVING CONTAINERS

Starting or Stopping a Container

When you create a container, its default state is disabled. By enabling a container, you start the processes of the containerized application in the compute resource. Hosts are then able to communicate with the container as with a web application.

1. In the Satellite web UI, navigate to **Containers** > **All Containers**.
2. From the list of existing containers, select the container that you want, and then click **Power On**. To stop the container, click **Power Off**.

Committing a Container

When you launch a container from an image, a writable layer is added on top of this image. Committing a container creates an image layer that stores the status of that container. Every time you commit a container a new image layer is added to store your changes. The container is committed to the repository of the original image. For example, if the container is based on an image pulled from the Docker Hub, the committed changes are pushed back to the Docker Hub.

1. In the Satellite web UI, navigate to **Containers** > **All Containers**.
2. From the list of existing containers, select the container that you want, and then click **Commit**.
3. Enter a repository name, for example **`user/my-rhel-image`**.

4. Assign a tag to the image.

5. Enter your contact information.

6. Enter an informative comment about the commit.

7. Click **Submit**.

Removing a Container

1. In the Satellite web UI, navigate to **Containers** > **All Containers**.

2. From the list of existing containers, select the container that you want, and then click **Delete**.

3. In the alert window, click **OK** to confirm the deletion.

APPENDIX A. INITIALIZATION SCRIPT FOR PROVISIONING EXAMPLES

If you have not followed the examples in the Red Hat Satellite 6 Content Management Guide, you can use the following initialization script to create an environment for provisioning examples.

Create a script file (**`sat6-content-init.sh`**) and include the following:

```
#!/bin/bash

MANIFEST=$1

# Import the content from Red Hat CDN
hammer organization create --name "ACME" --label "ACME" \
--description "Our example organization for managing content."
hammer subscription upload --file ~/$MANIFEST --organization "ACME"
hammer repository-set enable \
--name "Red Hat Enterprise Linux 7 Server (RPMs)" \
--releasever "7Server" --basearch "x86_64" \
--product "Red Hat Enterprise Linux Server" --organization "ACME"
hammer repository-set enable \
--name "Red Hat Enterprise Linux 7 Server (Kickstart)" \
--releasever "7Server" --basearch "x86_64" \
--product "Red Hat Enterprise Linux Server" --organization "ACME"
hammer repository-set enable \
--name "Red Hat Satellite Tools {ProductVersion} (for RHEL 7
Server) (RPMs)" \
--basearch "x86_64" --product "Red Hat Enterprise Linux Server" \
--organization "ACME"
hammer product synchronize --name "Red Hat Enterprise Linux Server"
\
--organization "ACME"

# Create our application life cycle
hammer lifecycle-environment create --name "Development" \
--description "Environment for ACME's Development Team" \
--prior "Library" --organization "ACME"
hammer lifecycle-environment create --name "Testing" \
--description "Environment for ACME's Quality Engineering Team" \
--prior "Development" --organization "ACME"
hammer lifecycle-environment create --name "Production" \
--description "Environment for ACME's Product Releases" \
--prior "Testing" --organization "ACME"

# Create and publish our Content View
hammer content-view create --name "Base" \
--description "Base operating system" \
--repositories "Red Hat Enterprise Linux 7 Server RPMs x86_64
```

```
7Server,Red Hat Satellite Tools {ProductVersion} for RHEL 7 Server
RPMs x86_64" \
--organization "ACME"
hammer content-view publish --name "Base" \
--description "Initial content view for our operating system" \
--organization "ACME"
hammer content-view version promote --content-view "Base" --version
1 \
--to-lifecycle-environment "Development" --organization "ACME"
hammer content-view version promote --content-view "Base" --version
1 \
--to-lifecycle-environment "Testing" --organization "ACME"
hammer content-view version promote --content-view "Base" --version
1 \
--to-lifecycle-environment "Production" --organization "ACME"
```

Set executable permissions on the script:

```
# chmod +x sat6-content-init.sh
```

Download a copy of your Subscription Manifest from the Red Hat Customer Portal and run the script on the manifest:

```
# ./sat6-content-init.sh manifest_98f4290e-6c0b-4f37-
ba79-3a3ec6e405ba.zip
```

This imports the necessary Red Hat content for the provisioning examples in this guide.

APPENDIX B. ADDITIONAL HOST PARAMETERS FOR

HAMMER CLI

This appendix provides some information on additional parameters for the **`hammer host create`** command.

B.1. COMMON INTERFACE PARAMETERS

These parameters are used with the **`--interface`** option for all provisioning types:

Parameter	**Description**
mac	MAC address for the interface
ip	IP address for the interface
type	The type of interface. For example: **`interface`**, **`bmc`**, or **`bond`**
name	The host name associated with this interface`
subnet_id	The subnet ID on the Satellite Server
domain_id	The domain ID on the Satellite Server

Parameter	Description
identifier	The device identifier. For example: **`eth0`**
managed	Boolean for managed interfaces. Set to **`true`** or **`false`**
primary	Boolean for primary interfaces. Managed hosts needs to have one primary interface. Set to **`true`** or **`false`**
provision	Boolean for whether to provision on this interface. Set to **`true`** or **`false`**
virtual	Boolean for whether the interface is a VLAN interface. Set to **`true`** or **`false`**

Use the following parameters if **`virtual`** is **`true`**:

Parameter	Description
tag	VLAN tag, this attribute has precedence over the subnet VLAN ID. Only for virtual interfaces.
attached_to	Identifier of the interface to which this

Parameter	Description
	interface belongs. For example: **`eth1`**.

Use the following parameters if **`type`** is **`bond`**:

Parameter	Description
mode	The bonding mode. One of **`balance-rr`**, **`active-backup`**, **`balance-xor`**, **`broadcast`**, **`802.3ad`**, **`balance-tlb`**, **`balance-alb`**
attached_devices	Identifiers of slave interfaces. For example: **`[eth1,eth2]`**
bond_options	Additional bonding options

Use the following parameters if **`type`** is **`bmc`**:

Parameter	Description
provider	The BMC provider. Only **`IPMI`** is supported
username	The username for the BMC device

B.2. EC2 PARAMETERS

Available parameters for **`--compute-attributes`**:

Parameter	Description
flavor_id	The EC2 flavor to use
image_id	The AMI ID of the image to use
availability_zone	The availability zone within the region of the EC2 provider
security_group_ids	The IDs for security groups to use
managed_ip	To utilize a public or private IP

B.3. LIBVIRT PARAMETERS

Available keys for **`--compute-attributes`**:

Parameter	Description
cpus	Number of CPUs

Parameter	Description
memory	Amount of memory in bytes
start	Boolean to start the machine

Available keys for **`--interface`**:

Parameter	Description
compute_type	Either **`bridge`** or **`network`**
compute_network / compute_bridge	Name of the network or physical interface
compute_model	The interface model. One of **`virtio`**, **`rtl8139`**, **`ne2k_pci`**, **`pcnet`**, or **`e1000`**

Available keys for **`--volume`**:

Parameter	Description
pool_name	The storage pool to store the volume

Parameter	Description
capacity	The capacity of the volume. For example: **`10G`**
format_type	The disk type. Either **`raw`** or **`qcow2`**

B.4. RED HAT OPENSTACK PLATFORM PARAMETERS

Available keys for **`--compute-attributes`**:

Parameter	Description
flavor_ref	The flavor to use
image_ref	The image to use
tenant_id	The tenant to use
security_groups	A list of security groups to use
network	The network to connect the instance

B.5. RED HAT VIRTUALIZATION PARAMETERS

Available keys for `--compute-attributes`:

Parameter	Description
cluster	The cluster ID to contain the host
template	The hardware profile to use
cores	The number of CPU cores to use
memory	The amount of memory in bytes
start	Boolean to start the machine

Available keys for `--interface`:

Parameter	Description
compute_name	The interface name. For example: `eth0`
compute_network	The network in the cluster to use

Available keys for `--volume`:

Parameter	Description
size_gb	Volume size in GB
storage_domain	The storage domain to use
bootable	Boolean to set the volume as bootable. Only one volume can be bootable

B.6. VMWARE INTERFACE PARAMETERS

Available keys for **`--compute-attributes`**:

Parameter	Description
cpus	Number of CPUs for the host
corespersocket	Number of cores per CPU socket. Applicable to hosts using hardware versions less than v10.
memory_mb	Amount of memory in MB
cluster	Cluster ID for the host

Parameter	Description
path	Path to folder to organize the host
guest_id	Guest OS ID
scsi_controller_type	ID of the VMware controller
hardware_version	VMware hardware version ID
start	Boolean to start the machine

Available keys for **`--interface`**:

Parameter	Description
compute_type	Type of the network adapter. One of **`VirtualVmxnet`**, **`VirtualVmxnet2`**, **`VirtualVmxnet3`**, **`VirtualE1000`**, **`VirtualE1000e`**, **`VirtualPCNet32`**.
compute_network	VMware network ID

Available keys for **`--volume`**:

Parameter	Description
datastore	The datastore ID
name	The name of the volume
size_gb	The size in GB
thin	Boolean value to enable thin provisioning
eager_zero	Boolean value to enable Eager Zero thick provisioning

APPENDIX C. PROVISIONING FIPS COMPLIANT HOSTS

Red Hat Satellite 6 supports provisioning hosts that comply with the National Institute of Standards and Technology's Security Requirements for Cryptographic Modules standard, reference number FIPS 140-2, referred to here as FIPS.

Red Hat Satellite 6 is not supported on a FIPS enabled host.

To enable the provisioning of hosts that are FIPS compliant, complete the following changes:

- Identify the relevant operating systems, locations, and organizations
- Create and enable the FIPS provisioning templates
- Change the provisioning password hashing algorithm
- Change the Puppet message digest algorithm
- Set the FIPS enabled parameter

When these changes are complete, the new provisioning templates will be associated with those operating systems, locations, and organizations you specify. When you provision a host to those operating systems, locations, and organizations, the host will have the FIPS-compliant settings applied. To confirm that these settings have been successful, complete the steps in Section C.6, “Verifying FIPS Mode is Enabled”.

Prerequisites

- Complete the configuration steps from the Authentication section in the *Hammer CLI Guide*. This allows you to run Hammer commands without providing your Satellite username and password each time.

C.1. IDENTIFYING THE RELEVANT OPERATING SYSTEMS, LOCATIONS, AND ORGANIZATIONS

Before creating the FIPS-compliant templates in Satellite, you must identify those locations, organizations and operating systems to which you want to deploy FIPS-compliant hosts. For example, if you will only deploy Red Hat Enterprise Linux 7 hosts as FIPS compliant, associate the template with only Red Hat Enterprise Linux 7.

1. List all locations.

   Example

   ```
   $ hammer location list
   ---|-----------------
   ID | NAME
   ---|-----------------
   ```

```
2  | Default Location
---|-----------------
```

Note the value in the **NAME** column of those locations to which you want to deploy FIPS-compliant hosts.

2. List all organizations.

Example

```
---|----------------------|----------------------|------------
ID | NAME                 | LABEL                | DESCRIPTION
---|----------------------|----------------------|------------
1  | Default Organization | Default_Organization |
2  | Sales                | Sales_Department     |
---|----------------------|----------------------|------------
```

Note the value in the **NAME** column of those organizations to which you want to deploy FIPS-compliant hosts.

3. List all operating systems.

Example

```
$ hammer os list
---|-----------------|--------------|-------
ID | TITLE           | RELEASE NAME | FAMILY
---|-----------------|--------------|-------
2  | RedHat 6.6      |              | Redhat
3  | RedHat 7.1      |              | Redhat
1  | RedHat 7.2      |              | Redhat
4  | RedHat 6.7      |              | Redhat
---|-----------------|--------------|-------
```

Note the value in the **TITLE** column of those operating systems to which you want to deploy FIPS-compliant hosts.

C.2. CREATING AND ENABLING THE FIPS PROVISIONING TEMPLATES

The FIPS provisioning templates are provided in a git repository. In this procedure you import them into the Satellite environment, then associate them with the desired operating systems, locations, and organizations.

1. On the Satellite Server, clone the git repository containing the FIPS enabled templates, then change into the repository's directory.

   ```
   $ git clone https://github.com/RedHatSatellite/satellite6-
   fips-client
   $ cd satellite6-fips-client
   ```

   This repository contains the following Embedded RuBy (ERB) templates. These are plain text files, which you can view to see in detail the configuration settings they contain.

   - **`Kickstart_Default_PXELinux_FIPS.erb`**
     - Updated PXELinux template
   - **`fips_packages.erb`**
     - Packages required by FIPS mode (for example, **`dracut-fips`**)
   - **`Satellite_Kickstart_Default_FIPS.erb`**
     - Kickstart template with modifications to call the **`fips_packages`** snippet
   - **`puppet.conf.erb`**
     - Updated **`puppet.conf`** configuration file with updated (SHA256) message digest algorithm

2. Add the *PXELinux FIPS* template.

   ```
   $ hammer template create  --name "Kickstart Default PXELinux
   FIPS" \
     --file Kickstart_Default_PXELinux_FIPS.erb  \
   ```

```
  --locations LOCATIONS \
  --organizations ORGANIZATION \
  --operatingsystems OS \
  --type PXELinux
```

Replace the placeholder values **LOCATIONS**, **ORGANIZATION**, and **OS** with the values you noted in Section C.1, “Identifying the Relevant Operating Systems, Locations, and Organizations”. If any value contains non-aphabetical characters, enclose the value in quotation marks (").

The message **Config template created** indicates success.

Example

```
$ hammer template create  --name "Kickstart Default PXELinux
FIPS" \
  --file Kickstart_Default_PXELinux_FIPS.erb \
  --locations "Default Location" \
  --organizations "Default Organization","Sales" \
  --operatingsystems "RedHat 6.6","RedHat 7.1","RedHat
7.2","RedHat 6.7" \
  --type PXELinux
```

3. Add the *Satellite Kickstart Default FIPS* template.

```
$ hammer template create  --name "Satellite Kickstart Default
FIPS" \
  --file Satellite_Kickstart_Default_FIPS.erb  \
  --locations LOCATIONS \
  --organizations ORGANIZATION \
  --operatingsystems OS \
  --type provision
```

Replace the placeholder values **LOCATIONS**, **ORGANIZATION**, and **OS** with the values you noted in Section C.1, “Identifying the Relevant Operating Systems, Locations, and Organizations”. If any value contains non-aphabetical characters, enclose the value in quotation marks (").

The message **`Config template created`** indicates success.

Example

```
$ hammer template create  --name "Satellite Kickstart Default
FIPS" \
  --file Satellite_Kickstart_Default_FIPS.erb  \
  --locations "Default Location" \
  --organizations "Default Organization","Sales" \
  --operatingsystems "RedHat 6.6","RedHat 7.1","RedHat
7.2","RedHat 6.7" \
  --type provision
```

4. Add the *FIPS Packages* snippet.

```
$ hammer template create  --name "fips_packages" \
  --file fips_packages.erb \
  --locations LOCATIONS \
  --organizations ORGANIZATION \
  --type snippet
```

Replace the placeholder values *LOCATIONS* and *ORGANIZATION* with the values you noted in Section C.1, “Identifying the Relevant Operating Systems, Locations, and Organizations”. If any value contains non-aphabetical characters, enclose the value in quotation marks (").

The message **`Config template created`** indicates success.

Example

```
$ hammer template create  --name "fips_packages" \
  --file fips_packages.erb \
  --locations "Default Location" \
  --organizations "Default Organization","Sales" \
  --type snippet
```

5. Update the default Puppet configuration snippet.

   ```
   $ hammer template update --name puppet.conf \
     --file puppet.conf.erb  \
     --type snippet
   ```

   The message **`Config template created`** indicates success.

6. Update the Operating System Object to use the new templates.

   Now that the new FIPS templates have been added to Satellite, they must be set as *default* templates for the desired operating system.

   a. Identify the IDs of the *Satellite Kickstart Default FIPS* and *Kickstart Default PXELinux FIPS* templates.

      Example

      ```
      $ hammer template list
      ---|----------------------------------------|----------
      ID | NAME                                   | TYPE
      ---|----------------------------------------|----------
      41 | redhat_register                        | snippet
      42 | saltstack_minion                       | snippet
      53 | Kickstart Default PXELinux FIPS        | PXELinux
      46 | Satellite Kickstart Default            | provision
      48 | Satellite Kickstart Default Finish     | finish
      54 | Satellite Kickstart Default FIPS       | provision
      47 | Satellite Kickstart Default User Data | user_data
      50 | subscription_manager_registration      | snippet
      29 | UserData default                       | user_data
      30 | WAIK default PXELinux                  | PXELinux
      ---|----------------------------------------|----------
      ```

      In this example, the IDs are 54 and 53 respectively. These IDs are installation specific.

   b. Specify the FIPS templates as default.

```
$ hammer os set-default-template --config-template-id
TEMPLATE \
--id OS
```

Replace the placeholders *TEMPLATE* and *OS* with the IDs of the FIPS templates, and the desired operating system, noted earlier. Repeat this command for every combination of FIPS template and operating system. It does not accept a comma-separated list of values.

In this example, the FIPS templates are set as default for Red Hat Enterprise Linux 7.2, identified in an earlier example as ID 1.

Example

```
$ hammer os set-default-template --config-template-id 54
--id 1
$ hammer os set-default-template --config-template-id 53
--id 1
```

C.3. CHANGE THE PROVISIONING PASSWORD HASHING ALGORITHM

This sets the password hashing algorithm used in provisioning to SHA256. This configuration setting must be applied for each operating system you want to deploy as FIPS compliant.

Note

This is required **ONLY** if Red Hat Satellite 6 was upgraded from Satellite 6.1. Satellite 6.4 uses SHA256 by default.

1. Identify the Operating System IDs.

Example

```
$ hammer os list
---|-----------------|--------------|-------
ID | TITLE           | RELEASE NAME | FAMILY
---|-----------------|--------------|-------
2  | RedHat 6.6      |              | Redhat
3  | RedHat 7.1      |              | Redhat
1  | RedHat 7.2      |              | Redhat
4  | RedHat 6.7      |              | Redhat
---|-----------------|--------------|-------
```

2. Update each operating system's password hash value.

```
$ hammer os update --title OS \
  --password-hash SHA256
```

Repeat this command for each of the desired operating systems, using the matching value in the **`TITLE`** column. It does not accept a comma-separated list of values.

Example

```
$ hammer os update --title "RedHat 7.2" \
  --password-hash SHA256
```

C.4. SWITCHING TO A FIPS COMPLIANT MESSAGE ALGORITHM FOR PUPPET

On the Satellite Server, all external Capsule Servers, and **all** existing hosts, configure Puppet to use the SHA256 message digest algorithm.

Edit the **`/etc/puppet/puppet.conf`** file, adding the line **`digest_algorithm = sha256`** in the **`[main]`** stanza.

Note

This change will be overwritten on every upgrade of Satellite, so needs to be reapplied afterward.

Because the Puppet message digest algorithm is changed on the Satellite Server and all Capsule Servers, it must also be changed on **all** hosts, including those that are not FIPS compliant.

In the event of a message digest algorithm mismatch, the client will download its facts again. This will result in a noticeable increased load on the Satellite Server or external Capsule Servers.

C.5. SETTING THE FIPS ENABLED PARAMETER

To provision a FIPS compliant host, the FIPS templates require a parameter named *fips_enabled* to be set to **`true`**. If this is not set to **`true`**, or is absent, the FIPS specific changes will not be applied. This parameter can be specified when provisioning an individual host, or set for a hostgroup. Retrospectively enabling FIPS compliance on a host is outside the scope of this guide and likely to cause problems.

To set this parameter when provisioning a host, append **`--parameters fips_enabled=true`** to the Hammer command.

To set this parameter on an existing host group, use the Hammer sub-command **`set-parameter`**. For more information, see the output of the command **`hammer hostgroup set-parameter --help`**. Any host provisioned to this hostgroup will inherit the *fips_enabled* parameter from the hostgroup.

Example

```
$ hammer hostgroup set-parameter --name fips_enabled \
 --value true \
 --hostgroup prod_servers
```

C.6. VERIFYING FIPS MODE IS ENABLED

To verify these FIPS compliance changes have been successful, you must provision a host and check its configuration.

1. Deploy a host using the FIPS templates, ensuring that parameter named *fips_enabled* is set to **`true`**.
2. Log in to the new host as a root-equivalent account.
3. Enter the command **`cat /proc/sys/crypto/fips_enabled`**. A value of **`1`** confirms that FIPS mode is enabled.

APPENDIX D. BUILDING CLOUD IMAGES FOR RED HAT SATELLITE

Use this section to build and register images to Red Hat Satellite.

You can use a preconfigured Red Hat Enterprise Linux KVM guest QCOW2 image:

- RHEL 7.4 KVM Guest Image
- RHEL 6.9 KVM Guest Image

These images contain **`cloud-init`**. To function properly, they must use ec2-compatible metadata services for provisioning an SSH key.

Note

For the KVM guest images:

- The **`root`** account in the image is disabled, but **`sudo`** access is granted to a special user named **`cloud-user`**.

- There is no **root** password set for this image.

  The **root** password is locked in **/etc/shadow** by placing **!!** in the second field.

If you want to create custom Red Hat Enterprise Linux images, see Creating a Red Hat Enterprise Linux 7 Image and Creating a Red Hat Enterprise Linux 6 Image.

D.1. CREATING CUSTOM RED HAT ENTERPRISE LINUX IMAGES

Prerequisites:

- Use a Linux host machine to create an image. In this example, we use a Red Hat Enterprise Linux 7 Workstation.
- Use **virt-manager** on your workstation to complete this procedure. If you create the image on a remote server, connect to the server from your workstation with **virt-manager**.
- A Red Hat Enterprise Linux 7 or 6 ISO file (see Red Hat Enterprise Linux 7.4 Binary DVD or Red Hat Enterprise Linux 6.9 Binary DVD).

For more information about installing a Red Hat Enterprise Linux Workstation, see Red Hat Enterprise Linux 7 Installation Guide.

Before you can create custom images, install the following packages:

- Install **libvirt**, **qemu-kvm** and graphical tools:

```
[root@host]# yum install virt-manager virt-viewer libvirt
qemu-kvm
```

- Install the following command line tools:

```
[root@host]# yum install virt-install libguestfs-tools-c
```

Note

In the following procedures, enter all commands with the **`[root@host]#`** prompt on the workstation that hosts the **`libvirt`** environment.

D.2. CREATING A RED HAT ENTERPRISE LINUX 7 IMAGE

Use this section to create an image in the QCOW2 format using a Red Hat Enterprise Linux 7 ISO file.

1. Using your web browser, download the Red Hat Enterprise Linux binary ISO file to a temporary location, for example, the **`Downloads`** directory.
2. Copy the Red Hat Enterprise Linux binary ISO file to the **`/var/lib/libvirt/images/`** directory.

```
[root@host]# cp ~/home/user/Downloads/rhel-server-7.4-x86_64-dvd.iso /var/lib/libvirt/images/
```

3. Verify that **virtbr0** is the virtual bridge:

```
[root@host]# ip a
```

4. Start **`libvirtd`**:

```
[root@host]# systemctl start libvirtd
```

5. Navigate to the **`/var/lib/libvirt/images/`** directory:

```
[root@host]# cd /var/lib/libvirt/images/
```

6. Prepare the QEMU image:

```
[root@host]# qemu-img create -f qcow2 rhel7.qcow2 8G
```

7. Start the installation using **`virt-install`**. Use the following example as a guide:

```
[root@host]# virt-install --virt-type qemu --name rhel7 --ram
2048 \
--cdrom rhel-server-7.4-x86_64-dvd.iso \
--disk rhel7.qcow2,format=qcow2 \
--network=bridge:virbr0 --graphics vnc,listen=0.0.0.0 \
--noautoconsole --os-type=linux --os-variant=rhel7
```

Note

For GUI users, if the instance does not launch automatically, enter the **`virt-manager`** command to view the console:

```
[root@host]# virt-manager
```

8. Follow the steps of the Red Hat Enterprise Linux installation wizard.

   a. For the installation source, add an HTTP link to your repository in Red Hat Satellite, for example **`satellite.example.com/pub/export/RHEL7/content/dist/rhel/server/7/7Server/x86_64/os/`**

   b. For the type of devices your installation uses, select **Auto-detected installation media**.

   c. For the type of installation destination, select **Local Standard Disks**.

   d. For other storage options, select **Automatically configure partitioning**.

   e. For software selection, select **Minimal Install**.

   f. Set the network interface to **ON** to ensure the interface activates on system start.

   g. Enter a host name, and click **Apply**.

   h. Enter a **`root`** password.

9. When the installation completes, reboot the instance and log in as the root user.

10. Confirm that the network interface is up and that the IP address is assigned:

    ```
    # ip a
    ```

11. Confirm that the hostname is correct:

    ```
    # hostname
    ```

12. Create a **`/etc/NetworkManager/conf.d/XX-cloud-image.conf`** file where *XX* is a two-digit number that indicates order of precedence. Add the following contents to the file:

    ```
    [main]
    dns=none
    ```

13. Proceed to Configuring a Host for Registration.

D.3. CREATING A RED HAT ENTERPRISE LINUX 6 IMAGE

Use this section to create an image in the QCOW2 format using a Red Hat Enterprise Linux 6 ISO file.

1. Start the installation using **`virt-install`**:

    ```
    [root@host]# qemu-img create -f qcow2 rhel6.qcow2 4G
    [root@host]# virt-install --connect=qemu:///system
    --network=bridge:virbr0 \
    --name=rhel6 --os-type linux --os-variant rhel6 \
    --disk path=rhel6.qcow2,format=qcow2,size=10,cache=none \
    --ram 4096 --vcpus=2 --check-cpu --accelerate \
    --hvm --cdrom=rhel-server-6.8-x86_64-dvd.iso
    ```

    This launches an instance and starts the installation process.

Note

If the instance does not launch automatically, enter the **`virt-viewer`** command to view the console:

```
[root@host]# virt-viewer rhel6
```

2. Set up the virtual machines as follows:

   a. At the initial Installer boot menu, select the **Install or upgrade an existing system** option.

   b. Select the appropriate **Language** and **Keyboard** options.

   c. When prompted about which type of devices your installation uses, select **Basic Storage Devices**.

   d. Select a **`hostname`** for your device. The default host name is **`localhost.localdomain`**.

   e. Set a root password.

   f. Based on the space on the disk, select the type of installation.

   g. Select the **Basic Server** install, which includes an SSH server.

3. Reboot the instance and log in as the **`root`** user.

4. Update the **`/etc/sysconfig/network-scripts/ifcfg-eth0`** file so it only contains the following values:

```
TYPE=Ethernet
DEVICE=eth0
ONBOOT=yes
BOOTPROTO=dhcp
NM_CONTROLLED=no
```

5. Restart the service network:

```
# service network restart
```

6. Proceed to Configuring a Host for Registration.

D.4. CONFIGURING A HOST FOR REGISTRATION

Red Hat Enterprise Linux virtual machines register to Customer Portal Subscription Management by default. You must update each virtual machine configuration so that they receive updates from the correct Satellite Server or Capsule Server.

Prerequisites

- Hosts must be using the following Red Hat Enterprise Linux version:
  - 6.4 or later
  - 7.0 or later
- All architectures of Red Hat Enterprise Linux are supported (i386, x86_64, s390x, ppc_64).
- Ensure that a time synchronization tool is enabled and runs on the Satellite Servers, any Capsule Servers, and the hosts.
  - For Red Hat Enterprise Linux 6:

    ```
    # chkconfig ntpd on; service ntpd start
    ```

  - For Red Hat Enterprise Linux 7:

    ```
    # systemctl enable chronyd; systemctl start chronyd
    ```

- Ensure that the daemon **`rhsmcertd`** is enabled and running on the hosts.
  - For Red Hat Enterprise Linux 6:

    ```
    # chkconfig rhsmcertd on; service rhsmcertd start
    ```

  - For Red Hat Enterprise Linux 7:

```
# systemctl start rhsmcertd
```

To Configure a Host for Registration:

1. Take note of the fully qualified domain name (FQDN) of the Satellite Server or Capsule Server, for example *server.example.com*.

2. On the host, connect to a terminal on the host as the root user

3. Install the consumer RPM from the Satellite Server or Capsule Server to which the host is to be registered. The consumer RPM updates the content source location of the host and allows the host to download content from the content source specified in Red Hat Satellite.

```
# rpm -Uvh http://server.example.com/pub/katello-ca-consumer-
latest.noarch.rpm
```

D.5. REGISTERING A HOST

Prerequisites

- Ensure that an activation key that is associated with the appropriate content view and environment exists for the host. For more information, see Managing Activation Keys in the *Content Management Guide*. By default, an activation key has the **`auto-attach`** function enabled. The feature is commonly used with hosts used as hypervisors.

- Ensure that the version of the **`subscription-manager`** utility is 1.10 or higher. The package is available in the standard Red Hat Enterprise Linux repository.

  - On the Red Hat Enterprise Linux Workstation, connect to a terminal as the root user.

  - Register the host using Red Hat Subscription Manager:

```
# subscription-manager register --org="My_Organization"
--activationkey="MyKey"
```

Note

You can use the **`--environment`** option to override the content view and life cycle environment defined by the activation key. For example, to register a host to the content view "MyView" in a "Development" life cycle environment:

```
# subscription-manager register --org="My_Organization" \
--environment=Development/MyView \
--activationkey="MyKey"
```

Note

For Red Hat Enterprise Linux 6.3 hosts, the release version defaults to Red Hat Enterprise Linux 6 Server and must point to the 6.3 repository.

1. On Red Hat Satellite, select **Hosts** > **Content Hosts**.
2. Select the name of the host that needs to be changed.
3. In the **Content Host Content** section click the edit icon to the right of **Release Version**.
4. Select "6.3" from the **Release Version** drop-down menu.
5. Click **Save**.

D.6. INSTALLING THE KATELLO AGENT

Use the following procedure to install the Katello agent on a host registered to Satellite 6. The **`katello-agent`** package depends on the gofer package that provides the **`goferd service`**. This service must be enabled so that the Red Hat Satellite Server or Capsule Server can provide information about errata that are applicable for content hosts.

Prerequisites

The **`Satellite Tools`** repository must be enabled, synchronized to the Red Hat Satellite Server, and made available to your hosts as it provides the required packages. For more information about enabling Satellite Tools, see Installing the Katello Agent in *Managing Hosts*.

To Install the Katello Agent

1. Install the **`katello-agent`** RPM package using the following command:

   ```
   # yum install katello-agent
   ```

2. Ensure goferd is running:

   ```
   # systemctl start goferd
   ```

D.7. INSTALLING THE PUPPET AGENT

Use this section to install and configure the Puppet agent on a host. When you have correctly installed and configured the Puppet agent, you can navigate to **Hosts** > **All hosts** to list all hosts visible to Red Hat Satellite Server.

1. Install the Puppet agent RPM package using the following command:

   ```
   # yum install puppet
   ```

2. Configure the puppet agent to start at boot:

   On Red Hat Enterprise Linux 6:

   ```
   # chkconfig puppet on
   ```

   On Red Hat Enterprise Linux 7:

```
# systemctl enable puppet
```

D.8. COMPLETING THE RED HAT ENTERPRISE LINUX 7 IMAGE

1. Update the system:

   ```
   # yum update
   ```

2. Install the **`cloud-init`** packages:

   ```
   # yum install cloud-utils-growpart cloud-init
   ```

3. Open the **`/etc/cloud/cloud.cfg`** configuration file:

   ```
   # vi /etc/cloud/cloud.cfg
   ```

4. Under the heading **`cloud_init_modules`**, add:

   ```
   - resolv-conf
   ```

   The **`resolv-conf`** option automatically configures the **`resolv.conf`** when an instance boots for the first time. This file contains information related to the instance such as **`nameservers`**, **`domain`** and other options.

5. Open the **`/etc/sysconfig/network`** file:

   ```
   # vi /etc/sysconfig/network
   ```

6. Add the following line to avoid problems accessing the EC2 metadata service:

   ```
   NOZEROCONF=yes
   ```

7. Un-register the virtual machine so that the resulting image does not contain the same subscription details for every instance cloned based on it:

   ```
   # subscription-manager repos --disable=*
   # subscription-manager unregister
   ```

8. Power off the instance:

   ```
   # poweroff
   ```

9. On your Red Hat Enterprise Linux Workstation, connect to the terminal as the root user and navigate to the **`/var/lib/libvirt/images/`** directory:

   ```
   [root@host]# cd /var/lib/libvirt/images/
   ```

10. Reset and clean the image using the **`virt-sysprep`** command so it can be used to create instances without issues:

    ```
    [root@host]# virt-sysprep -d rhel7
    ```

11. Reduce image size using the **`virt-sparsify`** command. This command converts any free space within the disk image back to free space within the host:

    ```
    [root@host]# virt-sparsify --compress rhel7.qcow2 rhel7-cloud.qcow2
    ```

    This creates a new **`rhel7-cloud.qcow2`** file in the location where you enter the command.

D.9. COMPLETING THE RED HAT ENTERPRISE LINUX 6 IMAGE

1. Update the system:

```
# yum update
```

2. Install the **cloud-init** packages:

```
# yum install cloud-utils-growpart cloud-init
```

3. Edit the **/etc/cloud/cloud.cfg** configuration file and under **cloud_init_modules** add:

```
- resolv-conf
```

The **resolv-conf** option automatically configures the **resolv.conf** configuration file when an instance boots for the first time. This file contains information related to the instance such as **nameservers**, **domain**, and other options.

4. To prevent network issues, create the **/etc/udev/rules.d/75-persistent-net-generator.rules** file as follows:

```
# echo "#" > /etc/udev/rules.d/75-persistent-
net-generator.rules
```

This prevents **/etc/udev/rules.d/70-persistent-net.rules** file from being created. If **/etc/udev/rules.d/70-persistent-net.rules** is created, networking might not function properly when booting from snapshots (the network interface is created as "eth1" rather than "eth0" and IP address is not assigned).

5. Add the following line to **/etc/sysconfig/network** to avoid problems accessing the EC2 metadata service:

```
NOZEROCONF=yes
```

6. Un-register the virtual machine so that the resulting image does not contain the same subscription details for every instance cloned based on it:

```
# subscription-manager repos --disable=*
# subscription-manager unregister
# yum clean all
```

7. Power off the instance:

```
# poweroff
```

8. On your Red Hat Enterprise Linux Workstation, log in as root and reset and clean the image using the **virt-sysprep** command so it can be used to create instances without issues:

```
[root@host]# virt-sysprep -d rhel6
```

9. Reduce image size using the **virt-sparsify** command. This command converts any free space within the disk image back to free space within the host:

```
[root@host]# virt-sparsify --compress rhel6.qcow2 rhel6-
cloud.qcow2
```

This creates a new **rhel6-cloud.qcow2** file in the location where you enter the command.

Note

You must manually resize the partitions of instances based on the image in accordance with the disk space in the flavor that is applied to the instance.

D.10. NEXT STEPS

- Repeat the procedures for every image that you want to provision with Satellite.
- Move the image to the location where you want to store for future use.

LEGAL NOTICE