



Red Hat Satellite 6.5-beta

Managing Hosts

A guide to managing hosts in a Red Hat Satellite 6 environment.

Red Hat Satellite 6.5-beta Managing Hosts

A guide to managing hosts in a Red Hat Satellite 6 environment.

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

Legal Notice

Copyright © 2019 Red Hat, Inc.

The text of and illustrations in this document are licensed by Red Hat under a Creative Commons Attribution–Share Alike 3.0 Unported license ("CC-BY-SA"). An explanation of CC-BY-SA is available at

<http://creativecommons.org/licenses/by-sa/3.0/>

. In accordance with CC-BY-SA, if you distribute this document or an adaptation of it, you must provide the URL for the original version.

Red Hat, as the licensor of this document, waives the right to enforce, and agrees not to assert, Section 4d of CC-BY-SA to the fullest extent permitted by applicable law.

Red Hat, Red Hat Enterprise Linux, the Shadowman logo, JBoss, OpenShift, Fedora, the Infinity logo, and RHCE are trademarks of Red Hat, Inc., registered in the United States and other countries.

Linux ® is the registered trademark of Linus Torvalds in the United States and other countries.

Java ® is a registered trademark of Oracle and/or its affiliates.

XFS ® is a trademark of Silicon Graphics International Corp. or its subsidiaries in the United States and/or other countries.

MySQL ® is a registered trademark of MySQL AB in the United States, the European Union and other countries.

Node.js ® is an official trademark of Joyent. Red Hat Software Collections is not formally related to or endorsed by the official Joyent Node.js open source or commercial project.

The OpenStack ® Word Mark and OpenStack logo are either registered trademarks/service marks or trademarks/service marks of the OpenStack Foundation, in the United States and other countries and are used with the OpenStack Foundation's permission. We are not affiliated with, endorsed or sponsored by the OpenStack Foundation, or the OpenStack community.

All other trademarks are the property of their respective owners.

Abstract

This guide describes how to configure and work with hosts in a Red Hat Satellite environment. Before continuing with this workflow you must have successfully installed a Red Hat Satellite 6 Server and any required Capsule Servers.

Table of Contents

CHAPTER 1. INTRODUCTION	4
1.1. UNDERSTANDING HOSTS	4
1.2. UNDERSTANDING HOST MANAGEMENT	4
CHAPTER 2. MANAGING HOSTS	5
2.1. BROWSING HOSTS	5
2.2. HOST STATUS TYPES	5
2.3. HOST OVERVIEW	6
2.4. CREATING A HOST	7
2.5. SETTING A HOST'S SYSTEM PURPOSE	9
2.6. EDITING A HOST'S SYSTEM PURPOSE	9
2.7. ASSIGNING ANSIBLE ROLES TO EXISTING HOSTS	10
2.8. RUNNING ANSIBLE ROLES ON A HOST	11
2.9. REGISTRATION	11
2.9.1. Configuring a Host for Registration	11
2.9.2. Registering a Host	13
2.9.3. Registering an Atomic Host	14
2.9.4. Installing the Katello Agent	15
2.9.5. Installing Tracer	16
2.9.6. Installing and Configuring the Puppet Agent	17
2.9.7. Registering Hosts to Satellite 6 Using The Bootstrap Script	19
2.9.8. Advanced Bootstrap Script Configuration	23
2.10. CHANGING A MODULE STREAM FOR A HOST	28
2.11. CREATING A HOST GROUP	28
2.12. CHANGING THE GROUP OF A HOST	30
2.13. CHANGING THE ENVIRONMENT OF A HOST	30
2.14. ADDING AN ANSIBLE ROLE TO A HOST GROUP	30
2.15. RUNNING ANSIBLE ROLES ON A HOST GROUP	30
2.16. MANAGING HOSTS	31
2.17. ASSIGNING A HOST TO A SPECIFIC ORGANIZATION	31
2.18. ASSIGNING A HOST TO A SPECIFIC LOCATION	32
2.19. CONFIGURING AN ADDITIONAL NETWORK INTERFACE	32
2.19.1. Adding a Physical Interface	33
2.19.2. Adding a Virtual Interface	34
2.19.3. Adding a Bonded Interface	35
2.19.4. Adding a Baseboard Management Controller (BMC) Interface	36
2.20. REMOVING A HOST	37
CHAPTER 3. GENERATING HOST MONITORING REPORTS	39
CHAPTER 4. CONFIGURING HOST COLLECTIONS	40
4.1. CREATING A HOST COLLECTION	40
4.2. CLONING A HOST COLLECTION	40
4.3. REMOVING A HOST COLLECTION	40
4.4. ADDING A HOST TO A HOST COLLECTION	41
4.5. REMOVING A HOST FROM A HOST COLLECTION	41
4.6. ADDING CONTENT TO A HOST COLLECTION	41
4.6.1. Adding Packages to a Host Collection	41
4.6.2. Adding Errata to a Host Collection	42
4.7. REMOVING CONTENT FROM A HOST COLLECTION	42
4.8. CHANGING THE LIFE CYCLE ENVIRONMENT OR CONTENT VIEW OF A HOST COLLECTION	43

CHAPTER 5. RUNNING JOBS ON HOSTS	44
5.1. ESTABLISHING A SECURE CONNECTION FOR REMOTE COMMANDS	44
5.2. SETTING UP KERBEROS AUTHENTICATION FOR REMOTE EXECUTION	46
5.3. CONFIGURING AND RUNNING REMOTE JOBS	47
5.3.1. Setting up Job Templates	47
5.3.2. Executing Jobs	49
5.3.3. Monitoring Jobs	50
5.3.4. Creating Advanced Templates	51
5.4. CONFIGURING GLOBAL SETTINGS	52
5.4.1. Choosing a Capsule for Remote Execution	53
5.5. DELEGATING PERMISSIONS FOR REMOTE EXECUTION	53
CHAPTER 6. DISCOVERING BARE-METAL HOSTS ON SATELLITE	55
6.1. NETWORK CONFIGURATION FOR PXE-BASED DISCOVERY	55
6.2. CONFIGURING THE SATELLITE DISCOVERY PLUG-IN	56
6.2.1. Deploying the Satellite Discovery Image	56
6.2.2. Configuring PXE-booting	56
6.2.3. Reviewing Global Discovery Settings	57
6.3. CONFIGURING THE SATELLITE CAPSULE SERVER DISCOVERY PLUG-IN	58
6.3.1. Configuring Discovery Subnets	58
6.3.2. Using Hammer with the Discovery Plug-in	58
6.3.3. Reviewing User Permissions	59
6.4. PROVISIONING DISCOVERED HOSTS	59
6.4.1. Manually Provisioning Hosts	59
6.4.2. Decommissioning Discovered Hosts	59
6.4.3. Automatically Provisioning Hosts	60
6.4.4. Scoped Search Syntax	61
6.4.5. Host Name Patterns	61
6.4.6. Using the Discovery Plug-in on the Command Line	62
6.5. EXTENDING THE DISCOVERY IMAGE	62
6.6. TROUBLESHOOTING SATELLITE DISCOVERY	63
CHAPTER 7. INTEGRATING RED HAT SATELLITE AND ANSIBLE TOWER	65
7.1. ADDING SATELLITE SERVER TO ANSIBLE TOWER AS A DYNAMIC INVENTORY ITEM	65
7.2. CONFIGURING PROVISIONING CALLBACK FOR A HOST	66
CHAPTER 8. SAMPLE SCENARIOS	69
8.1. SIMPLE SCENARIO	69
8.1.1. Creating the Host	69
8.1.2. Registering the Host	70
8.1.3. Running a Job on the Host	72
APPENDIX A. TEMPLATE WRITING REFERENCE	73
A.1. WRITING ERB TEMPLATES	73
A.2. TROUBLESHOOTING ERB TEMPLATES	74
A.3. SATELLITE SPECIFIC FUNCTIONS AND VARIABLES	74
APPENDIX B. HOST MANAGEMENT WITHOUT GOFERD	80
B.1. PREREQUISITES	80
B.2. CONFIGURING HOST MANAGEMENT WITHOUT GOFERD AS THE SYSTEM DEFAULT	80
B.3. LIMITATIONS WITH HAMMER	80

CHAPTER 1. INTRODUCTION

This chapter describes what a host is, types of hosts, and actions you can perform on them.

1.1. UNDERSTANDING HOSTS

A host is any Red Hat Enterprise Linux system that Red Hat Satellite manages. Hosts may be physical or virtual. Virtual hosts may be deployed on any platform supported by Red Hat Satellite, including: KVM, VMware vSphere, OpenStack, Amazon EC2, Rackspace Cloud Services, Google Compute Engine, or in a Docker container.

1.2. UNDERSTANDING HOST MANAGEMENT

Red Hat Satellite enables host management at scale, including monitoring, provisioning, remote execution, configuration management, software management, and subscription management. You can manage your hosts via the Red Hat Satellite web UI, or the command line. For more information on managing hosts, see [Chapter 2, *Managing Hosts*](#).

CHAPTER 2. MANAGING HOSTS

This chapter describes creating, registering, managing, and removing hosts. This includes changing the group, and environment of a host, configuring an additional network interface, assigning the host to a specific organization, and location.

2.1. BROWSING HOSTS

The Satellite Server web UI provides an opportunity to browse all hosts recognized by the Satellite Server. Navigate to **Hosts** tab at the top of the screen to open the drop-down menu with the following items:

- **All Hosts** - a list of all hosts recognized by the Satellite Server.
- **Discovered Hosts** - a list of bare-metal hosts detected on the provisioning network by the Discovery plug-in.
- **Content Hosts** - a list of hosts which manage tasks related to content and subscriptions.
- **Host Collections** - a list of user-defined collections of hosts used for bulk actions such as Errata Installation.

To search for a host, type in the **Search** field, and use an asterisk (*) to perform a partial string search. For example, if searching for a content host named **dev-node.example.com**, click the **Content Hosts** page, type **dev-node*** in the **Search** field. Alternatively, ***node*** will also find the content host **dev-node.example.com**.



WARNING




Satellite Server will be listed as a host itself even if it is not self-registered. Do not delete the Satellite Server from the list of hosts.

2.2. HOST STATUS TYPES

Each host recognized by the Satellite Server is assigned a status type in accordance with the most recent action performed on or upcoming changes to be applied to that host. Navigate to **Hosts** → **All hosts** to view the status of each host. The following table outlines the status types to which hosts can be assigned:

Table 2.1. Host Status Types

Icon	Status	Description
------	--------	-------------

Icon	Status	Description
	Error	An error has been detected on the host. If you hover the mouse over the error icon, a tooltip showing the actual reason of the error will appear. You can see a more detailed report of issues by clicking on the host.
	Warning	The host has been configured, but no reports have been collected for that host over the last reporting interval.
	OK	There are no pending actions on the host, no pending changes, and no errors over the last reporting interval.

2.3. HOST OVERVIEW

The host overview page displays information about a given host and the connection between the host and the installer. To view the host overview page, select **Hosts** → **All hosts**, then click the name of a host.

Details

The details bar contains a row of buttons that provide shortcuts to more information about the host, and tabs that display summaries of important details and events.

- **Audits:** a page containing audit entries for the current host.
- **Facts:** a page containing a list of facts for the current host. This button is only available after the installer has collected facts from the host.
- **Reports:** a page containing a list of reports for the current host. This button is only available after the installer has collected reports from the host.
- **YAML:** a page containing details about the host in YAML format, such as its IP address, MAC address, name, and values of parameters that have been applied to the host.
- **Properties:** a list of general details about the host, such as its IP address, MAC address, and the operating system entry that has been applied to the host.
- **Metrics:** a table showing a summary of all events reported for the host.
- **Templates:** a list of all provisioning templates currently accessible by the host. The provisioning templates include in this list are automatically configured in accordance with the operating system entry applied to the host.
- **NICs:** a table showing detailed information on NICs configured for the host.

Host Actions

Click each of these buttons to perform common actions on the host.

- **Schedule Remote Job:** allows running jobs on the host. For more information on running jobs see [Chapter 5, Running Jobs on Hosts](#).
- **Boot disk:** a menu that allows you to select the boot disk for the host. For more information on creating a boot ISO for a host see [Creating New Hosts with PXE-less Provisioning](#) in the *Red Hat Satellite Provisioning Guide*.
- **Edit:** opens the host details page which allows you to configure settings for the host. Note that the installer configures all the settings automatically and normally no manual configurations are required.
- **Build:** flags the host to be provisioned on the next host boot. Note that the installer manages all aspects of the provisioning process and normally there is no need to provision hosts manually.
- **Delete:** deletes the host from the user interface.

Host Graphs

The host overview page contains two graphs that display the status of recent Puppet runs executed on the host.

- **Runtime:** tracks two data points: **Config Retrieval** and **Runtime**. The **Config Retrieval** data point represents the amount of time taken to collect information about the host during a given Puppet run, and the **Runtime** data point represents the amount of time required to execute the Puppet run. Both data points are measured in seconds.
- **Resources:** tracks the number of actions performed on the host during a Puppet run. The categories displayed in this graph are identical to those displayed in the **Reports** page, and are measured using the number of actions in each category.

2.4. CREATING A HOST

The following procedure describes how to create a host in Red Hat Satellite.

To Create a Host:

1. Click **Hosts > Create Host**.
2. On the **Host** tab, enter the required details.
3. Click the **Ansible Roles** tab, and from the **Ansible Roles** list, select one or more roles that you want to add to the host. Use the **arrow** icon to manage the roles that you add or remove.
4. On the **Puppet Classes** tab, select the Puppet classes you want to include.
5. On the **Interfaces** tab:
 - a. For each interface, click **Edit** in the **Actions** column and configure the following settings as required:
 - **Type** — For a Bond or BMC interface, use the **Type** list and select the interface type.
 - **MAC address** — Enter the MAC address.

- **DNS name** — Enter the DNS name that is known to the DNS server. This is used for the host part of the FQDN.
 - **Domain** — Select the domain name of the provisioning network. This automatically updates the **Subnet** list with a selection of suitable subnets.
 - **IPv4 Subnet** — Select an IPv4 subnet for the host from the list.
 - **IPv6 Subnet** — Select an IPv6 subnet for the host from the list.
 - **IPv4 address** — If IP address management (IPAM) is enabled for the subnet, the IP address is automatically suggested. Alternatively, you can enter an address. The address can be omitted if provisioning tokens are enabled, if the domain does not manage DNS, if the subnet does not manage reverse DNS, or if the subnet does not manage DHCP reservations.
 - **IPv6 address** — If IP address management (IPAM) is enabled for the subnet, the IP address is automatically suggested. Alternatively, you can enter an address.
 - **Managed** — Select this check box to configure the interface during provisioning to use the Capsule provided DHCP and DNS services.
 - **Primary** — Select this check box to use the DNS name from this interface as the host portion of the FQDN.
 - **Provision** — Select this check box to use this interface for provisioning. This means TFTP boot will take place using this interface, or in case of image based provisioning, the script to complete the provisioning will be executed through this interface. Note that many provisioning tasks, such as downloading RPMs by **anaconda**, Puppet setup in a **%post** script, will use the primary interface.
 - **Virtual NIC** — Select this check box if this interface is not a physical device. This setting has two options:
 - **Tag** — Optionally set a VLAN tag. If unset, the tag will be the VLAN ID of the subnet.
 - **Attached to** — Enter the device name of the interface this virtual interface is attached to.
- b. Click **OK** to save the interface configuration.
 - c. Optionally, click **Add Interface** to include an additional network interface. See [Section 2.19, “Configuring an Additional Network Interface”](#) for details.
 - d. Press **Submit** to apply the changes and exit.
6. On the **Operating System** tab, enter the required details. For Red Hat operating systems, select **Synced Content** for **Media Selection**. If you want to use non Red Hat operating systems, select **All Media**, then select the installation media from the **Media Selection** list. You can select a partition table from the drop-down list or enter a custom partition table in the **Custom partition table** field. You cannot specify both.
 7. On the **Parameters** tab, click **Add Parameter** to add any parameter variables that you want to pass to job templates at run time. This includes all Puppet Class, Ansible playbook parameters and host parameters that you want to associate with the host. To use a parameter variable with an Ansible job template, you must add a **Host Parameter**.

8. On the **Additional Information** tab, enter additional information about the host.
9. Click **Submit** to complete your provisioning request.

2.5. SETTING A HOST'S SYSTEM PURPOSE

When you create a host with a Red Hat Enterprise Linux 8 operating system, you can set the system purpose attributes for the host. These system purpose attributes define what subscriptions to attach automatically on host creation. For more information about system purpose, see [System Purpose](#) in the *Installing and deploying RHEL 8* guide.

Also, you can edit the system purpose attributes and automatically attach subscriptions after you create the host. For more information, see [Section 2.6, "Editing a Host's System Purpose"](#).

Procedure

To define system purpose attributes when creating a host, perform the following steps:

1. In the Satellite web UI, navigate to **Hosts > Create Host**.
2. Click the **Parameters** tab.
3. In the **Host Parameters** area, enter in the following parameter names with the corresponding values. For the list of values, see [System Purpose](#) in the *Installing and deploying RHEL 8* guide.
 - **syspurpose_role**
 - **syspurpose_sla**
 - **syspurpose_usage**
 - **syspurpose_addons**
4. Click **Submit**.
5. Proceed to [Section 2.4, "Creating a Host"](#) to create a host.

2.6. EDITING A HOST'S SYSTEM PURPOSE

For a host with a Red Hat Enterprise Linux 8 operating system, you can edit the system purpose attributes. These system purpose attributes define what subscriptions to attach automatically to this host. For more information about system purpose, see [System Purpose](#) in the *Installing and deploying RHEL 8* guide.

Procedure

To edit the system purpose attributes for a host and automatically attach subscriptions, complete the following steps:

1. In the Satellite web UI, navigate to **Hosts > Content Hosts** and select the Red Hat Enterprise Linux 8 host that you want to edit.
2. In the lower left of the host window, navigate to the **System Purpose** area.
3. Click the **Edit** or **Remove** icon for the system purpose attributes that you want to edit, add, or remove.

4. Click **Save**.
5. Click the **Subscriptions** tab and select **Subscriptions**.
6. Click **Run Auto-Attach** to attach subscriptions to your host automatically.
7. Refresh the page to verify that the subscriptions list contains the correct subscriptions.

For the CLI Users

To edit the system purpose attributes for a host and automatically attach subscriptions, complete the following steps in the CLI of a content host:

1. Edit the required system purpose attributes. For example, set the usage type to **Production**, the role to **Red Hat Enterprise Linux Server**, and add the **addon** add on. For the list of values, see [System Purpose](#) in the *Installing and deploying RHEL 8* guide.

```
# syspurpose set-usage Production
# syspurpose set-role Red Hat Enterprise Linux Server
# syspurpose add-addons 'your_addon'
```

2. Verify the system purpose attributes for this host:

```
# syspurpose show
```

3. Automatically attach subscriptions to this host:

```
# subscription-manager attach --auto
```

4. Verify the system purpose status for this host:

```
# subscription-manager status
```

2.7. ASSIGNING ANSIBLE ROLES TO EXISTING HOSTS

You can use Ansible roles for remote management of Red Hat Enterprise Linux version 6.9 or later.

Before you can assign Ansible roles to hosts in Satellite, ensure that you import the roles to Satellite. For more information, see [Adding Red Hat Enterprise Linux System Roles](#) in the *Administering Red Hat Satellite*.

If you have a host that you want to add or modify Ansible roles for that host, complete the following procedure:

1. In the Satellite web UI, navigate to **Hosts > All Hosts**, and on the host that you want to add an Ansible role to, click **Edit**.
2. Select the **Ansible Roles** tab, and in the **All items** list, search for the roles that you want to add.
3. Select the roles that you want to add, and click the arrow icon to move the roles to the **Selected items** list.
4. Click **Submit** to save your changes.

When you assign Ansible roles to hosts, you can then use Ansible for remote execution. For more information, see [Section 5.1, “Establishing a Secure Connection for Remote Commands”](#).

Overriding Parameter Variables

On the **Parameters** tab, click **Add Parameter** to add any parameter variables that you want to pass to job templates at run time. This includes all Ansible playbook parameters and host parameters that you want to associate with the host. To use a parameter variable with an Ansible job template, you must add a **Host Parameter**.

2.8. RUNNING ANSIBLE ROLES ON A HOST

Before you begin, you must import Ansible roles in Satellite and assign the roles to your hosts. For more information, see [Section 2.7, “Assigning Ansible Roles to Existing Hosts”](#).

To Run an Ansible Role:

1. In the Satellite web UI, navigate to **Hosts > All Hosts**, and select the check box of the host that contains the Ansible role you want to run.
2. From the **Select Actions** list, select **Play Ansible roles**.

On the Run Ansible Roles page, you can view the status of your Ansible job. To rerun a job, click the **Rerun** button.

2.9. REGISTRATION

This section shows you how to register hosts to Satellite Server or Capsule Server. There are two main methods for registering a host:

- Download and install the consumer RPM (*server.example.com/pub/katello-ca-consumer-latest.noarch.rpm*) and then run subscription manager. This method is suited for freshly installed hosts. See [Section 2.9.1, “Configuring a Host for Registration”](#) and [Section 2.9.2, “Registering a Host”](#) for more information.
- Download and run the bootstrap script (*server.example.com/pub/bootstrap.py*). This method is suited for both freshly installed hosts and hosts that have been previously registered, for example, to Satellite 5 or another Satellite 6. See [Section 2.9.7, “Registering Hosts to Satellite 6 Using The Bootstrap Script”](#) for more information.

Hosts registered to the Satellite Server via Red Hat Subscription Manager, which can occur either during the post phase of a kickstart or through the terminal, will appear on the **Content Hosts** page accessible through **Hosts > Content Hosts**. Hosts provisioned by Satellite Server appear on the **Hosts** page accessible through **Hosts > All hosts**.

2.9.1. Configuring a Host for Registration

Red Hat Enterprise Linux hosts register to the Customer Portal Subscription Management by default. You must update each host configuration so that they receive updates from the correct Satellite Server or Capsule Server.

Supported Host Operating Systems

Hosts must use the following Red Hat Enterprise Linux version:

- 5.7 or later

- 6.1 or later*
- 7.0 or later



NOTE

Red Hat Enterprise Linux versions 6.1, 6.2 and 6.3 GA require **subscription-manager** and related packages to be updated manually. For more information, see <https://access.redhat.com/solutions/1256763>

Supported Architectures

All architectures of Red Hat Enterprise Linux are supported:

- i386
- x86_64
- s390x
- ppc_64

Prerequisites

- Ensure that the Satellite Servers, any Capsule Servers, and all hosts are synchronized with the same NTP server.
- Ensure that a time synchronization tool is enabled and running 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 start chronyd; systemctl enable chronyd
```

- Ensure that the daemon **rhsmcertd** is running on the hosts.

- For Red Hat Enterprise Linux 6:

```
# service rhsmcertd start
```

- For Red Hat Enterprise Linux 7:

```
# systemctl start rhsmcertd
```

The following procedure shows how to configure your host to register to Red Hat Satellite.

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*.

- In a terminal, connect to the host as the root user.
- 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
```



IMPORTANT

Any running Docker Daemons will be restarted.



NOTE

The RPM package is not signed, if required, you can add the `--nosignature` option to install the package. The `katello-ca-consumer-hostname-1.0-1.noarch.rpm` is an additional `katello-ca-consumer` RPM available that contains the server's host name. The `katello-ca-consumer-latest.noarch.rpm` rpm will always reflect the most recent version. Both serve the same purpose.

2.9.2. Registering a Host

Prerequisites

- Complete all steps in [Section 2.9.1, “Configuring a Host for Registration”](#).
- Ensure that an activation key associated with the appropriate Content View and environment exists for the host. If not, see [Managing Activation Keys](#) in the *Content Management Guide* for more information. 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 installed is 1.10 or later. The package is available in the standard Red Hat Enterprise Linux repository.

To Register Hosts:

- In a terminal, connect to the host as the root user.
- Clear any old host data related to Red Hat Subscription Manager (RHSM):

```
# subscription-manager clean
```

- Register the host using RHSM:

```
# subscription-manager register --org=your_org_name \
--activationkey=your_activation_key
```

Example 2.1. Command Output after Registration:

```
# subscription-manager register --org=MyOrg --
activationkey=TestKey-1
The system has been registered with id: 62edc0f8-855b-4184-b1b8-
```

72a9dc793b96

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=your_org_name \
--environment=Development/MyView \
--activationkey=your_activation_key
```

NOTE

For Red Hat Enterprise Linux 6.3 hosts, the release version defaults to Red Hat Enterprise Linux 6 Server and needs to be pointed to the 6.3 repository.

To Point Red Hat Enterprise Linux 6.3 to the Repository:

1. On Red Hat Satellite, select **Hosts** > **Content Hosts**.
2. Click 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**.

2.9.3. Registering an Atomic Host

Use the following procedure to register an Atomic Host with Subscription Manager.

1. Retrieve **katello-rhsm-consumer** from Satellite server:

```
[root@atomic_client ~]# wget
http://satellite.example.com/pub/katello-rhsm-consumer
```

2. Change the mode of **katello-rhsm-consumer** in order to make it executable:

```
[root@atomic_client ~]# chmod +x katello-rhsm-consumer
```

3. Run **katello-rhsm-consumer**:

```
[root@atomic_client ~]# ./katello-rhsm-consumer
```

Register with **Red Hat Subscription Manager**:

```
[root@atomic_client ~]# subscription-manager register
```

Because Atomic is functionally an appliance, do not install **katello-agent** on it.

2.9.4. Installing the Katello Agent

The following procedure shows how 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.

Note that **yum** is the only package management command that will update Satellite by triggering a Katello package upload. Avoid using **rpm** commands to install or update packages.

Prerequisites

Satellite version 6.1 and later require that you enable the **Red Hat Satellite Tools 6** repository. The **Red Hat Common** repositories are no longer used and are not compatible with Satellite version 6.1 and later.

The **Red Hat Satellite Tools 6** repository must be enabled, synchronized to the Red Hat Satellite Server and made available to your hosts as it provides the required packages.

To Verify the Red Hat Satellite Tools 6 Repository is Enabled:

1. Open the Satellite web UI, navigate to **Content > Red Hat Repositories** and click the **RPMS** tab.
2. Find and expand the **Red Hat Enterprise Linux Server** item.
3. Find and expand the **Red Hat Satellite Tools 6 (for RHEL *VERSION* Server) (RPMS)** item.
If the **Red Hat Satellite Tools 6** items are not visible, it may be because they are not included in the subscription manifest obtained from the Customer Portal. To correct that, log in to the Customer Portal, add these repositories, download the subscription manifest and import it into Satellite.
4. Ensure the **Enabled** check box beside the repository's name is selected. If not, select it.

Enable the **Red Hat Satellite Tools 6** repository for every supported major version of Red Hat Enterprise Linux running on your hosts.

To Install Katello Agent:

1. On the host, verify that the **rhel-version-server-satellite-tools-6-rpms** repository is enabled. If you registered the host using an activation key with auto-attach enabled, the repository is enabled automatically already.

```
# yum repolist enabled | grep -i rhel-version-server-satellite-  
tools-6-rpms
```

If the **rhel-version-server-satellite-tools-6-rpms** is not enabled, enable it using the following command:

```
# subscription-manager repos --enable=rhel-version-server-satellite-  
tools-6-rpms
```

2. Install the **katello-agent** RPM package using the following command:

```
# yum install katello-agent
```

3. Ensure the **goferd** service is running.

- On Red Hat Enterprise Linux 6, run the following command:

```
# service goferd start
```

- On Red Hat Enterprise Linux 7, run the following command:

```
# systemctl start goferd
```

2.9.5. Installing Tracer

This section describes how you can integrate Red Hat Satellite 6.5-beta with Tracer, and provides instructions on how to install Tracer and access Traces.

Tracer displays a list of services and applications that are outdated and require to be restarted. The list is labeled Traces and you can access this in the Satellite web UI.



NOTE

The integration of Tracer with Satellite Server is a Technology Preview feature. Technology Preview features are not fully supported under Red Hat Subscription Service Level Agreements (SLAs), may not be functionally complete, and are not intended for production use. However, these features provide early access to upcoming product innovations, enabling customers to test functionality and provide feedback during the development process.

Prerequisite

For your content host, enable and synchronize the Red Hat Satellite Tools 6.5-beta repository.

Procedure

Tracer is an optional component, therefore you must install it separately from the rest of the Katello host tools.

1. On the content host, install the **katello-host-tools-tracer** RPM package:

```
# yum install katello-host-tools-tracer
```

2. Enter the following command:

```
# katello-tracer-upload
```

3. In the Satellite web UI, navigate to **Hosts > All hosts**, and then click the required host name.
4. In the **Properties** tab, examine the **Properties** table, and then find the Traces item. If you cannot find a Traces item in the **Properties** table, then Tracer is not installed.

To Access Traces:

Traces is the output generated by Tracer in the Satellite web UI.

1. In the Satellite web UI, navigate to **Hosts > Content Hosts**, and then click the required host name.
2. Click the **Traces** tab to view Traces.

2.9.6. Installing and Configuring the Puppet Agent

This section describes how 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.

Prerequisites

The **rhel-7-server-satellite-tools-6-beta-rpms** repository must be enabled, synchronized to the Red Hat Satellite Server and made available to your hosts as it provides the required packages.

To Verify the rhel-7-server-satellite-tools-6-beta-rpms Repository is Enabled:

1. In the Satellite web UI, navigate to **Content > Red Hat Repositories**.
2. In the search field, enter **Red Hat Satellite Tools 6.4 for RHEL 7 Server RPMs**.
3. Select **Red Hat Satellite Tools 6.4 (for RHEL 7 Server) (RPMs)** and select the enable icon next to **x86_64**.

If the **Red Hat Satellite Tools 6.4 (for RHEL 7 Server) (RPMs)** items are not visible, it might be because they are not included in the subscription manifest obtained from the Customer Portal. To correct that, log in to the Customer Portal, add these repositories, download the subscription manifest and import it into Satellite.

To Install and Enable the Puppet Agent:

1. On the host, open a terminal console and log in as the **root** user.
2. Verify that the **rhel-7-server-satellite-tools-6-beta-rpms** repository is enabled, using the following command:

```
# yum repolist enabled | grep -i rhel-7-server-satellite-tools-6-beta-rpms
```

If the **rhel-7-server-satellite-tools-6-beta-rpms** is not enabled, enable it using the following command:

```
# subscription-manager repos \
--enable=rhel-7-server-satellite-tools-6-beta-rpms
```

3. Install the Puppet agent RPM package using the following command:

```
# yum install puppet
```

4. Configure the puppet agent to start at boot:

- a. On Red Hat Enterprise Linux 6:

```
# chkconfig puppet on
```

- b. On Red Hat Enterprise Linux 7:

```
# systemctl enable puppet
```

Prerequisites

The following conditions must be met before configuring the Puppet Agent:

- The host must be registered to the Red Hat Satellite Server.
- The Red Hat Satellite Tools 6 repository must be enabled.
- Puppet packages must be installed on the host.

A Puppet environment is a collection of Puppet modules that can be associated with a host or a host group. Before you configure the Puppet agent environment, complete the following steps:

1. To find the host's Puppet environment, in the Satellite web UI, navigate to **Hosts > All Hosts** and view the **Environment** column in the host table.
2. To assign a Puppet environment to a host, navigate to **Hosts > All Hosts** and click **Edit** next to the selected host.
3. To list Puppet environments enabled on the Satellite Server, navigate to **Configure > Environments**. You can also inspect the `/etc/puppet/environments/` directory on the Satellite Server to find what Puppet modules and manifests are associated with Puppet environments.

For more information see the [Red Hat Satellite Puppet Guide](#).

To Configure the Puppet Agent:

1. Configure the Puppet agent by specifying the server and environment settings in the `/etc/puppet/puppet.conf` file:

```
# vi /etc/puppetlabs/puppet/puppet.conf
```

2. Set the **environment** parameter to the name of the Puppet environment to which the host belongs:

```
environment = My_Example_Org_Library
```

3. Run the Puppet agent on the host:

```
# puppet agent -t --server satellite.example.com
```

4. Sign the SSL certificate for the Puppet client in the Satellite Server web UI:
 - a. Log in to the Satellite Server web UI.
 - b. Select **Infrastructure > Capsules**.
 - c. Select **Certificates** from the drop-down menu to the right of the required Capsule.
 - d. Click **Sign** to the right of the required host.
 - e. Enter the **puppet agent** command again:

```
# puppet agent -t --server satellite.example.com
```

Assigning Context to a Host

When the Puppet agent is configured on the host, it appears in **All Hosts** but only when **Any Organization** is selected because the host is not assigned to an organization or location.

- To assign the host to an organization, see [Section 2.17, “Assigning a Host to a Specific Organization”](#).
- To assign the host to a location, see [Section 2.18, “Assigning a Host to a Specific Location”](#).

2.9.7. Registering Hosts to Satellite 6 Using The Bootstrap Script

The bootstrap script, which is included in 6.2 and later, can be used to register new hosts or migrate existing hosts to Satellite 6.

The bootstrap script handles content registration, product certificates, and Puppet configuration. The bootstrap script has the advantage of taking a Red Hat Enterprise Linux system, regardless of where it is registered (RHN, Satellite 5, SAM, RHSM), or if it is registered at all, and subscribing it to Satellite 6.

The bootstrap script package, **katello-client-bootstrap**, is installed by default on the Satellite Server’s base system and the script itself is installed in the **/var/www/html/pub/** directory to make it available to hosts. It can be accessed using a URL in the following form:

```
satellite6.example.com/pub/bootstrap.py
```

The script includes documentation in a readme file. To view the file on the Satellite CLI:

```
$ less /usr/share/doc/katello-client-bootstrap-version/README.md
```

Installing the Bootstrap Script on the Host:

As the script is only required once, and only for the **root** user, you can place it in **/root** and remove it after use, or place it in **/usr/local/sbin**. This example will use **/root**.

As **root**, install the bootstrap script on the host as follows:

1. Ensure you are in the correct directory. For example, to change to **/root**:

```
# cd
```

2. Download the script:

```
# curl -O http://satellite6.example.com/pub/bootstrap.py
```

This installs the script in the current directory.

3. Make the script executable:

```
# chmod +x bootstrap.py
```

4. To confirm that the script can now be run, view the usage statement as follows:

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py -h
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# ./bootstrap.py -h
```

5. Optionally, when the transition process is complete, remove the script:

```
# cd  
# rm bootstrap.py
```

Required Permissions The user requires certain permissions in order to run the bootstrap script. You can set these permissions using roles through the web UI or using the Hammer command line tool.

Using the web UI to set permissions for the bootstrap script:

1. Navigate to **Administer > Users**.
2. Select an existing user or create a new user especially for the purpose of running this script. Users are located under the **Username** field. A new pane will open with tabs to modify information about the selected user.
3. Click the **Roles** tab.
4. Select **Viewer** and **Edit hosts** from the **Roles** list. The selected roles should appear on the **Selected Items** list confirming that they have been selected.



WARNING

Be aware that the following role **Edit hosts** allows the user to edit and delete hosts as well as being able to add hosts. If this is not acceptable to your security policy, create a new role with the minimal permissions required by the bootstrap script. Create the *bootstrap* role using the Hammer command line tool, see [Using Hammer to set permissions for the bootstrap script](#), then assign this role alone, to the user who will be running the bootstrap script. Alternatively, create an appropriate role using the web UI, navigate to **Administer > Roles**. Subsequently, use the web UI to create a role and set up the appropriate filters.

5. Click **Submit**. The permissions required for the bootstrap script to run will now be set for the user you have specified.

Using Hammer to set permissions for the bootstrap script:

1. Create a role with the minimum permissions required by the bootstrap script. This example creates a role with the name *Bootstrap*. Modify this to provide a more specific name if you wish.


```

$ ROLE='Bootstrap'
$ hammer role create --name "$ROLE"
$ hammer filter create --role "$ROLE" --permissions
view_organizations
$ hammer filter create --role "$ROLE" --permissions view_locations
$ hammer filter create --role "$ROLE" --permissions view_domains
$ hammer filter create --role "$ROLE" --permissions view_hostgroups
$ hammer filter create --role "$ROLE" --permissions view_hosts
$ hammer filter create --role "$ROLE" --permissions
view_architectures
$ hammer filter create --role "$ROLE" --permissions view_ptables
$ hammer filter create --role "$ROLE" --permissions
view_operatingsystems
$ hammer filter create --role "$ROLE" --permissions create_hosts

```

2. Assign the new role to an existing user.

```
$ hammer user add-role --id user_id --role Bootstrap
```

There is also the option to create a new user and assign this new role to them. For more information on how to create users through Hammer see [Creating Users](#) in the Red Hat Satellite Hammer CLI Guide.

Running the Bootstrap Script

Prerequisites

- The bootstrap script is installed as described previously.
- You have an activation key for your hosts. For configuring activation keys, see [Managing Activation Keys](#) in the *Content Management Guide*.
- You have created a host group. For creating host groups, see [Section 2.11, “Creating a Host Group”](#).



NOTE

Puppet fails to retrieve the Puppet CA certificate while registering a host with a host group associated with a Puppet environment created inside a **Production** environment. To create a suitable Puppet environment to be associated with a host group, follow these steps:

1. Manually create a directory and change the owner:

```
# mkdir /etc/puppet/environments/example_environment
# chown apache
/etc/puppet/environments/example_environment
```

2. Navigate to **Configure** → **Environments** and click **Import environment from**. The button name will include the FQDN of the internal or external Capsule.
3. Choose the created directory and click **Update**.

Running the bootstrap script:

1. Enter the bootstrap command as follows with values suitable for your environment.



WARNING

The example in this section specifies the admin user. If this is not acceptable to your security policy, create a new role with the minimal permissions required by the bootstrap script that can be added to the appropriate user. This is achieved using the web UI or Hammer, see [Using the web UI to set permissions for the bootstrap script:](#) and [Using Hammer to set permissions for the bootstrap script:](#) for more information.

For the `--server` option, specify the FQDN name of Satellite Server or Capsule Server. For `--location`, `--organization`, and `--hostgroup` options, use quoted names, not labels, as arguments to the options. See [Section 2.9.8, “Advanced Bootstrap Script Configuration”](#) for advanced use cases.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \
--login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key
```

The script will prompt you for the password corresponding to the Satellite user name you entered with the `--login` option.

2. The script will run and send notices of progress to **stdout**. Watch for output prompting you to approve the certificate.

```
[NOTIFICATION], [2016-04-26 10:16:00], [Visit the UI and approve
this certificate via Infrastructure->Capsules]
[NOTIFICATION], [2016-04-26 10:16:00], [if auto-signing is disabled]
[RUNNING], [2016-04-26 10:16:00], [/usr/bin/puppet agent --test --
noop --tags no_such_tag --waitforcert 10]
```

The host will wait indefinitely until an administrator approves the Puppet certificate.

- a. In the web UI, navigate to **Infrastructure > Capsules**.

- b. Select **Certificates** to the right of the name of the Capsule corresponding to the FQDN given with **--server** option.
 - c. In the **Actions** column select **Sign** to approve the host's Puppet certificate.
 - d. Return to the host to see the remainder of the bootstrap process completing.
3. In the web UI, navigate to **Hosts > All hosts** and ensure that the host is connected to the correct host group.

If the Katello agent is not installed on the host, proceed to [Section 2.9.4, "Installing the Katello Agent"](#).

2.9.8. Advanced Bootstrap Script Configuration

The standard workflow of using the bootstrap script has been outlined in [Running the Bootstrap Script](#). This section covers some more examples.



WARNING

The examples in this section specify the admin user. If this is not acceptable to your security policy, create a new role with the minimal permissions required by the bootstrap script. See [Using the web UI to set permissions for the bootstrap script:](#) and [Using Hammer to set permissions for the bootstrap script:](#) for more information.

Migrating a host from one Satellite 6 to another Satellite 6.

Use the script with **--force**, and the script will remove the `katello-ca-consumer-*` packages from the old Satellite and install the `katello-ca-consumer-*` packages from the new Satellite.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \
--login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key \
--force
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key \
--force
```

Migrating a host from Red Hat Network (RHN) or Satellite 5 to Satellite 6.

The bootstrap script detects the presence of `/etc/syconfig/rhn/systemid` and a valid connection to RHN as an indicator that the system is registered to a legacy platform. The script then calls `rhn-classic-migrate-to-rhsm` to migrate the system from RHN. By default, the script does not delete the system's legacy profile due to auditing reasons. To remove the legacy profile, use `--legacy-purge` and use `--legacy-login` to supply an user account that has appropriate permissions to remove a profile. Enter the user account password when prompted.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \  
--login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--legacy-purge \  
--legacy-login rhn-user
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--legacy-purge \  
--legacy-login rhn-user
```

Registering a host to Satellite 6, omitting Puppet setup.

By default, the bootstrap script configures the host for content management and configuration management. If you have an existing configuration management system and do not want to install puppet on the host, use `--skip-puppet`.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \  
--login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--skip-puppet
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--skip-puppet
```

```
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--skip-puppet
```

Registering a host to Satellite 6 for content management only.

To register a system as a content host, and leave out the provisioning and configuration management functions, use **--skip-foreman**.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \  
--server satellite6.example.com \  
--organization="Example Organization" \  
--activationkey=activation_key \  
--skip-foreman
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --server satellite6.example.com \  
--organization="Example Organization" \  
--activationkey=activation_key \  
--skip-foreman
```

Changing the method the bootstrap script uses to download the consumer RPM.

By default, the bootstrap script uses HTTP to download the consumer RPM (*server.example.com/pub/katello-ca-consumer-latest.noarch.rpm*). In some environments, you might want to allow HTTPS only between the host and Satellite. Use **--download-method** to change the download method from HTTP to HTTPS.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \  
--login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--download-method https
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--download-method https
```

Providing the host's IP address to Satellite

On hosts with multiple interfaces or multiple IP addresses on one interface, you may need to override the auto-detection of the IP address and provide a specific IP address to Satellite. Use `--ip`.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \  
--login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--ip 192.x.x.x
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--ip 192.x.x.x
```

Enabling Remote Execution on the host.

Use `--rex` and `--rex-user` to enable remote execution and add the required SSH keys for the specified user. For example:

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \  
--login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--rex \  
--rex-user root
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \  
--server satellite6.example.com \  
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--rex \  
--rex-user root
```

Creating a domain for a host at registration time.

To create a host record, the DNS domain of a host needs to exist in Satellite prior to running script. If the domain does not exist, add it using **--add-domain**.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py \
--login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key \
--add-domain
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key \
--add-domain
```

Providing an arbitrary Fully Qualified Domain Name (FQDN) for the host.

1. If the host's host name is not an FQDN, or is not RFC compliant (containing a character such as an underscore), the script will fail at the host name validation stage. Set **create_new_host_when_facts_are_uploaded** and **create_new_host_when_report_is_uploaded** to false using **hammer**.

```
# hammer settings set \
--name create_new_host_when_facts_are_uploaded \
--value false
# hammer settings set \
--name create_new_host_when_report_is_uploaded \
--value false
```

2. Use **--fqdn** to specify the FQDN that will be reported to Satellite.

- On Red Hat Enterprise Linux 8, enter the following command:

```
# /usr/libexec/platform-python bootstrap.py --login=admin \
--server satellite6.example.com \
--location="Example Location" \
--organization="Example Organization" \
--hostgroup="Example Host Group" \
--activationkey=activation_key \
--fqdn node100.example.com
```

- On Red Hat Enterprise Linux 7, 6 or 5, enter the following command:

```
# bootstrap.py --login=admin \
--server satellite6.example.com \
```

```
--location="Example Location" \  
--organization="Example Organization" \  
--hostgroup="Example Host Group" \  
--activationkey=activation_key \  
--fqdn node100.example.com
```

2.10. CHANGING A MODULE STREAM FOR A HOST

If you have a Red Hat Enterprise Linux 8 host, you can modify the module stream for the repositories you install on a host. For example, after you create the host, you can enable, disable, install, update, and remove module streams from your host using Satellite using remote execution.

1. In the Satellite web UI, navigate to **Hosts > Content Hosts** and select the host that contains the modules you want to change.
2. Click the **Module Streams** tab.
3. From the **Available Module Streams** list, locate the module that you want to change. You can use the **Filter** field to refine the list entries. You can also use the **Filter Status** list to search for modules with a specific status.
4. On the module that you want to change, from the **Actions** list, select the change that you want to make to the module.
5. In the Job Invocation window, ensure that the job information is accurate. Change any details that you require, and then click **Submit**.

2.11. CREATING A HOST GROUP

If you create a high volume of hosts, many of the hosts can have common settings and attributes. Adding these settings and attributes for every new host is time consuming. If you use host groups, you can apply common attributes to hosts that you create.

A host group functions as a template for common host settings, containing many of the same details that you provide to hosts. When you create a host with a host group, the host inherits the defined settings from the host group. You can then provide additional details to individualize the host.

Host Group Hierarchy

You can also create a hierarchy of host groups. Aim to have one base level host group that represents all hosts in your organization and provide general settings, and then nested groups to provide specific settings. For example, you can have a base host level group that defines the operating system, and two nested host groups that inherit the base level host group:

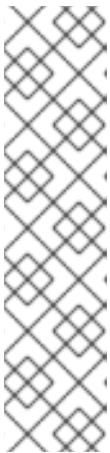
- **Hostgroup: Base** (Red Hat Enterprise Linux 7.2)
 - **Hostgroup: Webserver** (applies the `httpd` Puppet class)
 - **Host: webserver1.example.com** (web server)
 - **Host: webserver2.example.com** (web server)
 - **Hostgroup: Storage** (applies the `nfs` Puppet class)
 - **Host: storage1.example.com** (storage server)

- **Host: storage2.example.com** (storage server)
 - **Host: custom.example.com** (custom host)

In this example, all hosts use Red Hat Enterprise Linux 7.2 as their operating system because of their inheritance of the **Base** host group. The two web server hosts inherit the settings from the **Webserver** host group, which includes the **httpd** Puppet class and the settings from the **Base** host group. The two storage servers inherit the settings from the **Storage** host group, which includes the **nfs** Puppet class and the settings from the **Base** host group. The custom host only inherits the settings from the **Base** host group.

To Create a Host Group

1. In the Satellite web UI, navigate to **Configure > Host Groups** and click **Create Host Group**.
2. If you have an existing host group that you want to inherit attributes from, you can select a host group from the **Parent** list. If you do not, leave this field blank.
3. In the **Name** field, enter a name for the new host group.
4. Enter any further information that you want future hosts to inherit.
5. Click the **Ansible Roles** tab, and from the **Ansible Roles** list, select one or more roles that you want to add to the host. Use the **arrow** icon to manage the roles that you add or remove.
6. Click the additional tabs and add any details that you want to attribute to the host group.



NOTE

Puppet fails to retrieve the Puppet CA certificate while registering a host with a host group associated with a Puppet environment created inside a **Production** environment. To create a suitable Puppet environment to be associated with a host group, follow these steps:

1. Manually create a directory and change the owner:

```
# mkdir /etc/puppet/environments/example_environment
# chown apache
/etc/puppet/environments/example_environment
```

7. Click **Submit** to save the host group.

For CLI Users

Create the host group with the **hammer hostgroup create** command. For example:

```
# hammer hostgroup create --name "Base" \
--lifecycle-environment "Production" --content-view "Base" \
--environment "production" --content-source-id 1 \
--puppet-ca-proxy-id 1 --puppet-proxy-id 1 --domain "example.com" \
--subnet `ACME's Internal Network` --architecture "x86_64" \
--operatingsystem "RedHat 7.2" --medium-id 9 \
--partition-table "Kickstart default" --root-pass "p@55w0rd!" \
--locations "New York" --organizations "ACME"
```

2.12. CHANGING THE GROUP OF A HOST

The following steps show you how to change the group of a host.

1. Navigate to **Hosts > All hosts**.
2. Select the check box of the host you want to change.
3. From the **Select Action** menu at the upper right of the screen, select **Change Group**. A new option window will open.
4. From the **Host Group** menu, select the group that you want for your host.
5. Click **Submit**.

2.13. CHANGING THE ENVIRONMENT OF A HOST

The following steps show you how to change the environment of a host.

1. Navigate to **Hosts > All hosts**.
2. Select the check box of the host you want to change.
3. From the **Select Action** menu at the upper right of the screen, select **Change Environment**. A new option window will open.
4. From the **Environment** menu, select the new environment for your host.
5. Click **Submit**.

2.14. ADDING AN ANSIBLE ROLE TO A HOST GROUP

You can use Ansible roles for remote management of Red Hat Enterprise Linux version 6.9 or later.

Before you can assign Ansible roles to host groups in Satellite, ensure that you import the roles to Satellite. For more information, see [Adding Red Hat Enterprise Linux System Roles](#) in the *Administering Red Hat Satellite*.

If you have an existing host group that you want to add Ansible Roles to, complete the following steps:

1. In the Satellite web UI, navigate to **Configure > Host Groups**.
2. From the list of host groups, click the host group name that you want to add an Ansible Role to.
3. Click the **Ansible Roles** tab, and from the **Ansible roles** list, select one or more roles that you want to add to the host. Use the **arrow** icon to manage the roles that you add or remove.
4. Click **Submit** to save your changes.

2.15. RUNNING ANSIBLE ROLES ON A HOST GROUP

Before you begin, you must import Ansible roles in Satellite and assign the roles to your host group. For more information, see [Section 2.14, “Adding an Ansible Role to a Host Group”](#).

You must have at least one host in your host group.

To Run an Ansible Role:

1. In the Satellite web UI, navigate to **Configure > Host Groups**, and select the check box of the host group that contains the Ansible role you want to run.
2. From the **Actions** column for the host group, select the arrow icon to the right of the **Nest** button, and then select **Play Roles**.

On the Run Ansible Roles page, you can view the status of your Ansible job. To rerun a job, click the **Rerun** button.

2.16. MANAGING HOSTS

Hosts provisioned by Satellite are managed by default. When the host is set to managed, it is possible to configure additional host parameters from Satellite Server. These additional parameters are listed on the **Operating System** tab.

If you change any settings on the **Operating System** tab they will not take effect until you set the host to build and reboot it.

If there is a necessity to obtain reports about configuration management in systems using an operating system not supported by Satellite it is recommended to unmanage the host.

The following procedure shows how to switch a host between Managed and Unmanaged status.

1. Navigate to **Hosts > All hosts**.
2. Select the host.
3. Click **Edit**.
4. Click **Manage host** or **Unmanage host** to change the host's status.
5. Click **Submit** to save the changes.

2.17. ASSIGNING A HOST TO A SPECIFIC ORGANIZATION

The following steps show you how to assign a host to a specific organization. For general information about organizations and how to configure them, see [Managing Organizations](#) in the *Content Management Guide*.

1. Navigate to **Hosts > All hosts**.
2. Select the check box of the host you want to change.
3. From the **Select Action** menu at the upper right of the screen, select **Assign Organization**. A new option window will open.
4. Navigate to the **Select Organization** menu and choose the organization that you want to assign for your host. Select the check box **Fix Organization on Mismatch**.

**NOTE**

A mismatch happens if there is a resource associated with a host, such as a domain or subnet, and at the same time not associated with the organization you want to assign the host to. The option **Fix Organization on Mismatch** will add such a resource to the organization, and is therefore the recommended choice. The option **Fail on Mismatch**, on the other hand, will always result in an error message. For example, reassigning a host from one organization to another will fail, even if there is no actual mismatch in settings.

5. Click **Submit**.

2.18. ASSIGNING A HOST TO A SPECIFIC LOCATION

The following steps show you how to assign a host to a specific location. For general information about locations and how to configure them, see [Creating a Location](#) in the *Provisioning Guide*.

1. Navigate to **Hosts > All hosts**.
2. Select the check box of the host you want to change.
3. From the **Select Action** menu at the upper right of the screen, select **Assign Location**. A new option window will open.
4. Navigate to the **Select Location** menu and choose the location that you want for your host. Select the check box **Fix Location on Mismatch**.

**NOTE**

A mismatch happens if there is a resource associated with a host, such as a domain or subnet, and at the same time not associated with the organization you want to assign the host to. The option **Fix Organization on Mismatch** will add such a resource to the organization, and is therefore the recommended choice. The option **Fail on Mismatch**, on the other hand, will always result in an error message. For example, reassigning a host from one organization to another will fail, even if there is no actual mismatch in settings.

5. Click **Submit** to complete the assigning of the location to your host.

2.19. CONFIGURING AN ADDITIONAL NETWORK INTERFACE

Red Hat Satellite supports specifying multiple network interfaces for a single host. You can configure these interfaces when creating a new host as described in [Section 2.4, “Creating a Host”](#) or when editing an existing host.

There are several types of network interfaces that you can attach to a host. When adding a new interface, select one of:

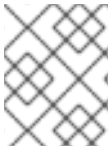
- **Interface**: Allows you to specify an additional physical or virtual interface. There are two types of virtual interfaces you can create. Use **VLAN** when the host needs to communicate with several (virtual) networks using a single interface, while these networks are not accessible to each other. Another type of virtual interface is **alias**, which is an additional IP address attached to an existing interface. See [Section 2.19.2, “Adding a Virtual Interface”](#), or [Section 2.19.1, “Adding a Physical Interface”](#) for details.

- **Bond:** Creates a bonded interface. NIC bonding is a way to bind multiple network interfaces together into a single interface that appears as a single device and has a single MAC address. This enables two or more network interfaces to act as one, simultaneously increasing the bandwidth and providing redundancy. See [Section 2.19.3, “Adding a Bonded Interface”](#) for details.
- **BMC:** Baseboard Management Controller (BMC) allows you to remotely monitor and manage physical state of machines. For more information about BMC, see [Enabling Power Management on Managed Hosts](#) in *Installing Satellite Server from a Connected Network* and for more information about configuring BMC interfaces, see [Section 2.19.4, “Adding a Baseboard Management Controller \(BMC\) Interface”](#).



NOTE

Additional interfaces have by default the **Managed** flag enabled, which means the new interface is configured automatically during provisioning by the DNS and DHCP Capsule Servers associated with the selected subnet. This requires a subnet with correctly configured DNS and DHCP Capsule Servers. If you use a kickstart method for host provisioning, configuration files are automatically created for managed interfaces in the post-installation phase at `/etc/sysconfig/network-scripts/ifcfg-$interface_id`.



NOTE

Virtual and bonded interfaces currently require a MAC address of a physical device. Therefore, the configuration of these interfaces works only on bare-metal hosts.

2.19.1. Adding a Physical Interface

The following steps show how to add an additional physical interface to a host.

To Add a Physical Interface:

1. Navigate to **Hosts** > **All hosts** to view available hosts.
2. Click **Edit** next to the host you want to edit.
3. On the **Interfaces** tab, click **Add Interface**.
4. Keep the **Interface** option selected in the **Type** menu.
5. Specify a **MAC address** of the additional interface. This setting is required.
6. Specify the device **Identifier**, for example `eth0` or `eth1.1`. Identifier is used for bonded interfaces (in the **Attached devices** field, see [To Add a Bonded Interface](#)), VLANs and aliases (in the **Attached to** field, see [To Add a Virtual Interface](#)).
7. Specify the **DNS name** associated with the host's IP address. Satellite saves this name in the Capsule Server associated with the selected domain (the "DNS A" field) and the Capsule Server associated with the selected subnet (the "DNS PTR" field). A single host can therefore have several DNS entries.
8. Select a domain from the **Domain** drop-down menu. To create and manage domains, navigate to **Infrastructure** > **Domains**.
9. Select a subnet from the **Subnet** drop-down menu. To create and manage subnets, navigate to **Infrastructure** > **Subnets**.

10. Specify the interface **IP address**. Managed interfaces with assigned DHCP Capsule Server require this setting for creating a DHCP lease. DHCP-enabled managed interfaces provide an automatic suggestion of IP address.
11. Decide if the interface will be managed. If the **Managed** check box is selected, the interface configuration is pulled from the associated Capsule Server during provisioning, and DNS and DHCP entries are created. If using kickstart provisioning, a configuration file is automatically created for the interface.
12. Select the **Virtual NIC** check box to create a virtual interface. See [Section 2.19.2, “Adding a Virtual Interface”](#) for details.
13. Click **OK** to save the interface configuration, and then click **Submit** to apply the changes to the host.

2.19.2. Adding a Virtual Interface

The following procedure shows how to configure an additional virtual interface for a host. This can be either a VLAN or an alias interface.

An alias interface is an additional IP address attached to an existing interface. Note that:

- An alias interface automatically inherits a MAC address from the interface it is attached to, therefore you can create an alias without specifying a MAC address.
- The interface must be specified in a subnet with boot mode set to **static**.

To Add a Virtual Interface:

1. Navigate to **Hosts** > **All hosts** to view available hosts.
2. Click **Edit** next to the host you want to edit.
3. On the **Interfaces** tab, click **Add Interface**.
4. Keep the **Interface** option selected in the **Type** menu.
5. Specify the general interface settings. The applicable configuration options are the same as for the physical interfaces described in [Section 2.19.1, “Adding a Physical Interface”](#). Specify **MAC address** for managed virtual interfaces so that the configuration files for provisioning are generated correctly. However, **MAC address** is not required for virtual interfaces that are not managed.

If creating a VLAN, specify ID in the form of **eth1.10** in the **Identifier** field. If creating an alias, use ID in the form of **eth1:10**.

6. Select the **Virtual NIC** check box. Additional configuration options specific to virtual interfaces are appended to the form:
 - **Tag**: You can specify tags per interface to provide a higher-level segmentation of the network. If left blank, managed interfaces inherit the tag from the VLAN ID of the associated subnet, given that this subnet has the VLAN ID specified. User-specified entries from this field are not applied on alias interfaces.
 - **Attached to**: Specify the identifier of the physical interface to which the virtual interface belongs, for example eth1. This setting is required.

- Click **OK** to save the interface configuration. Then click **Submit** to apply the changes to the host.

2.19.3. Adding a Bonded Interface

The following steps show how to configure a bonded interface for a host.

To Add a Bonded Interface:

- Navigate to **Hosts** > **All hosts** to view available hosts.
- Click **Edit** next to the host you want to edit.
- On the **Interfaces** tab, click **Add Interface**.
- Select **Bond** from the **Type** menu. Additional type-specific configuration options are appended to the form.
- Specify the general interface settings. The applicable configuration options are the same as for the physical interfaces described in [Section 2.19.1, “Adding a Physical Interface”](#). Bonded interfaces use IDs in the form of **bond0** in the **Identifier** field. It is sufficient if you specify a single MAC address in the **MAC address** field.
- Specify the configuration options specific to bonded interfaces:
 - **Mode:** Select the bonding mode that defines a policy for fault tolerance and load balancing. See [Table 2.2, “Bonding Modes Available in Red Hat Satellite”](#) for a brief description of individual bonding modes.
 - **Attached devices:** Specify a comma separated list of identifiers of attached devices. These can be physical interfaces or VLANs.
 - **Bond options:** Specify a space separated list of configuration options, for example **miimon=100**. There are several configuration options you can specify for the bonded interface, see [Red Hat Enterprise Linux 7 Networking Guide](#) for details.
- Click **OK** to save the interface configuration. Then click **Submit** to apply the changes to the host.

Table 2.2. Bonding Modes Available in Red Hat Satellite

Bonding Mode	Description
balance-rr	Transmissions are received and sent out sequentially on each bonded interface.
active-backup	Transmissions are received and sent out via the first available bonded interface. Another bonded interface is only used if the active bonded interface fails.
balance-xor	Transmissions are based on the selected hash policy. In this mode, traffic destined for specific peers will always be sent over the same interface.
broadcast	All transmissions are sent on all bonded interfaces.

Bonding Mode	Description
802.a3	Creates aggregation groups that share the same settings. Transmits and receives on all interfaces in the active group.
balance-tlb	The outgoing traffic is distributed according to the current load on each bonded interface.
balance-alb	Receive load balancing is achieved through Address Resolution Protocol (ARP) negotiation.

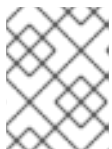
2.19.4. Adding a Baseboard Management Controller (BMC) Interface

This section describes how to configure a baseboard management controller (BMC) interface for a host that supports this feature.

Prerequisites

Ensure the following prerequisites are satisfied before proceeding:

- BMC is enabled on the Capsule Server. If required, see [To Enable BMC Power Management on an Existing Capsule Server](#).
- The `ipmitool` package is installed.
- You know the MAC address, IP address, and other details of the BMC interface on the host, and the appropriate credentials for that interface.



NOTE

You only need the MAC address for the BMC interface if the BMC interface is managed. This is so that it can create a DHCP reservation.

To Enable BMC Power Management on an Existing Capsule Server:

1. Use the `satellite-installer` routine to configure BMC power management on the Capsule Server by running the following command with the following options:

```
# satellite-installer --foreman-proxy-bmc=true \ --foreman-proxy-bmc-default-provider=ipmitool
```

2. Refresh the features for the Capsule Server.
 - a. Log in to the Satellite web UI, and navigate to **Infrastructure > Capsules**.
 - b. Identify the Capsule Server whose features you need to refresh. In the drop-down list on the right, click **Refresh**. The list of features in the **Features** column should now include BMC.

To Add a BMC Interface:

1. Navigate to **Hosts > All hosts** to view available hosts.
2. Click **Edit** next to the host you want to edit.
3. On the **Interfaces** tab, click **Add Interface**.
4. Select **BMC** from the **Type** menu. Type-specific configuration options are appended to the form.
5. Specify the general interface settings. The applicable configuration options are the same as for the physical interfaces described in [Section 2.19.1, “Adding a Physical Interface”](#).
6. Specify the configuration options specific to BMC interfaces:
 - **Username, Password:** Specify any authentication credentials required by BMC.
 - **Provider:** Specify the BMC provider.
7. Click **OK** to save the interface configuration, and then click **Submit** to apply the changes to the host.

2.20. REMOVING A HOST

The following procedure shows how to remove a host from Red Hat Satellite.

To Remove a Host:

1. Click **Hosts > All hosts** or **Hosts > Content Hosts**.
2. Select the hosts that you want to remove.
3. Click **Select Action** and select **Delete Hosts** from the drop-down menu.
4. Click **Submit** to remove the host from Red Hat Satellite permanently.



WARNING

If a host record that is associated with a virtual machine is deleted, the virtual machine will be deleted as well. To avoid deleting the virtual machine in this situation, disassociate the virtual machine from Satellite without removing it from the hypervisor.

To Disassociate A Virtual Machine from Satellite without Removing it from a Hypervisor

1. In the Satellite web UI, navigate to **Hosts > All Hosts** and select the check box to the left of the hosts to be disassociated.
2. From the **Select Action** drop-down menu, select the **Disassociate Hosts** button.
3. In the confirmation window:
 - a. Optionally, select the check box to keep the hosts for future action.

- b. Click **Submit** to save your changes.

CHAPTER 3. GENERATING HOST MONITORING REPORTS

You can use monitoring reports to query registered hosts in Satellite to obtain information such as host status, inventories, applicable errata and subscription details.

Satellite Server uses ERB syntax report templates, similar to provisioning templates and remote execution job templates, to generate host monitoring reports.

To view the host monitoring report templates, in the Satellite web UI, navigate to **Monitor > Report Templates**. You can create a template, or clone a template and edit the clone. For help with the template syntax, click a template and click the **Help** tab.

To schedule reports, configure a cron job.

Procedure

To generate a host monitoring report, complete the following steps:

1. In the Satellite web UI, navigate to **Monitor > Report Templates**.
2. To the right of the report template that you want to use, click **Generate**.
3. Optional: Apply search query filters. To view all available results, do not populate the filter field with any values.
4. Click **Submit**. A CSV file that contains the report is downloaded.

For CLI Users

To generate a report, complete the following steps:

1. List all available report templates:

```
# hammer report-template list
```

2. Generate a report:

```
# hammer report-template generate --id template ID
```

CHAPTER 4. CONFIGURING HOST COLLECTIONS

A host collection is a group of multiple content hosts. This feature enables you to perform the same action on multiple hosts at once. These actions can include the installation, removal, and update of packages and errata, change of assigned life cycle environment, and change of Content View. You can create host collections to suit your requirements, and those of your company. For example, group hosts in host collections by function, department, or business unit.

4.1. CREATING A HOST COLLECTION

The following procedure shows how to create host collections.

To Create a Host Collection:

1. Click **Hosts > Host Collections**.
2. Click **New Host Collection**.
3. Add the Name of the host collection.
4. Clear **Unlimited Content Hosts**, and enter the desired maximum number of hosts in the **Limit** field.
5. Add the Description of the host collection.
6. Click **Save**.

4.2. CLONING A HOST COLLECTION

The following procedure shows how to clone a host collection.

To Clone a Host Collection:

1. Click **Hosts > Host Collections**.
2. On the left hand panel, click the host collection you want to clone.
3. Click **Copy Collection**.
4. Specify a name for the cloned collection.
5. Click **Create**.

4.3. REMOVING A HOST COLLECTION

The following procedure shows how to remove a host collection.

To Remove a Host Collection:

1. Click **Hosts > Host Collections**.
2. Choose the host collection to be removed.
3. Click **Remove**. An alert box appears:

Are you sure you want to remove host collection *Host Collection Name*?

4. Click **Remove**.

4.4. ADDING A HOST TO A HOST COLLECTION

The following procedure shows how to add hosts to host collections.

Prerequisites

A host must be registered to Red Hat Satellite in order to add it to a Host Collection. Refer to [Section 2.9, “Registration”](#) for information on how to register a host.

To Add Hosts to a Host Collection:

1. Click **Hosts > Host Collections**.
2. Click the host collection where the host should be added.
3. On the **Hosts** tab, select the **Add** subtab.
4. Select the hosts to be added from the table and click **Add Selected**.

4.5. REMOVING A HOST FROM A HOST COLLECTION

The following procedure shows how to remove hosts from host collections.

To Remove Hosts from a Host Collection:

1. Click **Hosts > Host Collections**.
2. Choose the desired host collection.
3. On the **Hosts** tab, select the **List/Remove** subtab.
4. Select the hosts you want to remove from the host collection and click **Remove Selected**.

4.6. ADDING CONTENT TO A HOST COLLECTION

These steps show how to add content to host collections in Red Hat Satellite.

4.6.1. Adding Packages to a Host Collection

The following procedure shows how to add packages to host collections.

Prerequisites

- The content to be added should be available in one of the existing repositories or added prior to this procedure.
- Content should be promoted to the environment where the hosts are assigned.

To Add Packages to Host Collections:

1. Click **Hosts > Host Collections**.
2. Click the host collection where the package should be added.
3. On the **Collection Actions** tab, click **Package Installation, Removal, and Update**.
4. To update all packages, click the **Update All Packages** button to use the default method. Alternatively, select the drop-down icon to the right of the button to select a method to use. Selecting the **via remote execution - customize first** menu entry will take you to the **Job invocation** page where you can customize the action.
5. Select the **Package** or **Package Group** radio button as required.
6. In the field provided, specify the package or package group name. Then click:
 - **Install** — to install a new package using the default method. Alternatively, select the drop-down icon to the right of the button and select a method to use. Selecting the **via remote execution - customize first** menu entry will take you to the **Job invocation** page where you can customize the action.
 - **Update** — to update an existing package in the host collection using the default method. Alternatively, select the drop-down icon to the right of the button and select a method to use. Selecting the **via remote execution - customize first** menu entry will take you to the **Job invocation** page where you can customize the action.

4.6.2. Adding Errata to a Host Collection

The following procedure shows how to add errata to host collections.

Prerequisites

- The errata to be added should be available in one of the existing repositories or added prior to this procedure.
- Errata should be promoted to the environment where the hosts are assigned.

To Add Errata to a Host Collection:

1. Click **Hosts > Host Collections**.
2. Select the host collection where the errata should be added.
3. On the **Collection Actions** tab, click **Errata Installation**.
4. Select the errata you want to add to the host collection and click the **Install Selected** button to use the default method. Alternatively, select the drop-down icon to the right of the button to select a method to use. Selecting the **via remote execution - customize first** menu entry will take you to the **Job invocation** page where you can customize the action.

4.7. REMOVING CONTENT FROM A HOST COLLECTION

The following procedure shows how to remove packages from host collections.

To Remove Content from a Host Collection:

1. Click **Hosts > Host Collections**.
2. Click the host collection where the package should be removed.
3. On the **Collection Actions** tab, click **Package Installation, Removal, and Update**.
4. Select the **Package** or **Package Group** radio button as required.
5. In the field provided, specify the package or package group name.
6. Click the **Remove** button to remove the package or package group using the default method. Alternatively, select the drop-down icon to the right of the button and select a method to use. Selecting the **via remote execution - customize first** menu entry will take you to the **Job invocation** page where you can customize the action.

4.8. CHANGING THE LIFE CYCLE ENVIRONMENT OR CONTENT VIEW OF A HOST COLLECTION

The following procedure shows how to change the assigned life cycle environment or Content View of host collections.

To Change the Life Cycle Environment or Content View of a Host Collection:

1. Click **Hosts > Host Collection**.
2. Selection the host collection where the life cycle environment or Content View should be changed.
3. On the **Collection Actions** tab, click **Change assigned Life Cycle Environment or Content View**.
4. Select the life cycle environment to be assigned to the host collection.
5. Select the required Content View from the drop-down list.
6. Click **Assign**.



NOTE

The changes take effect in approximately 4 hours. To make the changes take effect immediately, on the host, enter the following command:

```
# subscription-manager refresh
```

You can use remote execution to run this command on multiple hosts at the same time.

CHAPTER 5. RUNNING JOBS ON HOSTS

Red Hat Satellite supports the ability to run jobs with arbitrary commands on hosts using shell scripts and to run Ansible tasks and playbooks. This is referred to as remote execution.

Remote execution is enabled by default on the Satellite Server, but must be enabled manually on Capsule Servers. For custom Ansible roles that you create, or roles that you download, the package containing the roles must be installed on the Capsule's base system where a task is executed, or a playbook is run. The roles must also be imported into Satellite from the Capsule where they are installed before Satellite can use them.

Communication occurs through the Capsule Server which means that Satellite Server does not require direct access to the target host, and can scale to control many hosts. Remote execution uses the SSH service which must be enabled and running on the target host. Ensure the Capsule has access to port 22 on the target hosts.

Satellite uses ERB syntax job templates similar to the provisioning templates. Several job templates for shell scripts and Ansible are included by default. See [Section 5.3.1, “Setting up Job Templates”](#).



NOTE

Any Capsule Server's base system is a client of Satellite Server's internal Capsule, and therefore this section applies to any type of host connected to Satellite Server, including Capsule Servers.

You can run jobs on multiple hosts at once, and you can use variables in your commands to suit your deployment. Variable values can be filled by host fact, Smart Class Parameter, Smart Variable, or even host parameter. In addition, you can specify custom values for templates when you run the command. See [Section 5.3.2, “Executing Jobs”](#).

By default, each Capsule is installed with the remote execution feature disabled. To enable remote execution, enter the following command:

```
# satellite-installer --scenario capsule \
  --enable-foreman-proxy-plugin-remote-execution-ssh
```

To verify that remote execution is enabled on the Capsule Server, in the web UI navigate to **Infrastructure > Capsules** and in the **Actions** column, to the right of the edit icon, select **Refresh features** and confirm the feature is listed.

By default, Satellite Server is configured to use remote execution rather than Katello Agent. These settings can be changed by first creating custom job templates and then selecting these new templates in the web UI by going to **Administer > Remote Execution Features**. For each action you want to change, select the label and then select the job template to use.

5.1. ESTABLISHING A SECURE CONNECTION FOR REMOTE COMMANDS

The SSH keys used for remote execution are created automatically when installing a Capsule and the settings are in the `/etc/foreman-proxy/settings.d/remote_execution_ssh.yml` file. They include the following options:

ssh_identity_file

File to load the SSH key from. By default, set to `/usr/share/foreman-proxy/.ssh/id_rsa_foreman_proxy`.

local_working_dir

Directory used on the Satellite or Capsule to run the scripts necessary for remote execution. By default, set to `/var/tmp`.

remote_working_dir

Directory on the client system that is used to execute the remote execution jobs. By default, set to `/var/tmp`.



NOTE

If the client system has `noexec` set for the `/var/` volume or file system, change the `remote_working_dir` as otherwise the remote execution job will fail since the script cannot be run.

If you must use an alternative directory, create the new directory, for example `new_place`, and then copy the SELinux context from the default directory. For example:

```
# chcon --reference=/var new_place
```

See [Maintaining SELinux Labels](#) in the *SELinux User's and Administrator's Guide* for more information on working with SELinux labels.

Distributing SSH Keys for Remote Execution

To use SSH keys for authenticating remote execution connections, distribute the public SSH key from a Capsule to its attached hosts that you want to manage. Ensure the SSH service is enabled and running on the hosts. Configure any network or host-based firewalls to enable access to port 22.

There are three ways to distribute the public key from a Capsule to target hosts:

- To distribute SSH keys manually, enter the following command on the Capsule:

```
# ssh-copy-id -i ~foreman-proxy/.ssh/id_rsa_foreman_proxy.pub
root@target.example.com
```

Where `target.example.com` is the host name of the target host. Repeat for each target host you want to manage.

To confirm the key was successfully copied to the target host, enter the following command on the Capsule:

```
# ssh -i ~foreman-proxy/.ssh/id_rsa_foreman_proxy
root@target.example.com
```

- To use the Satellite API to download the public key directly from the Capsule, enter the following command on each target host:

```
# curl https://capsule.example.com:9090/ssh/pubkey >>
~/.ssh/authorized_keys
```

Where `capsule.example.com` is the host name of the Capsule that the host is attached to.

- To include the public key in newly-provisioned hosts, modify for example the **Kickstart default finish** template to include the following line:

```
<%= snippet 'remote_execution_ssh_keys' %>
```

5.2. SETTING UP KERBEROS AUTHENTICATION FOR REMOTE EXECUTION

From Satellite 6.5-beta, you can use Kerberos authentication to establish an SSH connection for remote execution on Satellite hosts.

Prerequisites

Before you can use Kerberos authentication for remote execution on Red Hat Satellite, you must set up a Kerberos server for identity management and ensure that you complete the following prerequisites:

- Enroll Satellite Server on the Kerberos server
- Enroll the Satellite target host on the Kerberos server
- Configure and initialize a Kerberos user account for remote execution
- Ensure that the **foreman-proxy** user on Satellite has a valid Kerberos ticket granting ticket

To set up Satellite to use Kerberos authentication for remote execution on hosts, complete the following steps:

1. Before installing the **tfm-rubygem-net-ssh-krb** package, you must temporarily set SELinux to **permissive** until [Red Hat bug 1541481](#) is resolved:

```
# setenforce 0
```

2. To install the **tfm-rubygem-net-ssh-krb** package, enter the following command:

```
# yum install tfm-rubygem-net-ssh-krb
```

3. To install and enable Kerberos authentication for remote execution, enter the following command:

```
# satellite-installer --scenario satellite \
  --foreman-proxy-plugin-remote-execution-ssh-ssh-kerberos-auth true
```

4. Set SELinux to **enforcing**:

```
# setenforce 1
```

5. To edit the default user for remote execution, in the Satellite web UI, navigate to **Administer > Settings** and click the **RemoteExecution** tab. In the **remote_execution_ssh_user** row, edit the second column and add the user name for the Kerberos account.
6. Navigate to **remote_execution_effective_user** and edit the second column to add the user name for the Kerberos account.

To confirm that Kerberos authentication is ready to use, run a remote job on the host.

5.3. CONFIGURING AND RUNNING REMOTE JOBS

Any command that you want to apply to a remote host must be defined as a job template. After you have defined a job template you can execute it multiple times.

5.3.1. Setting up Job Templates

Satellite provides default job templates that you can use for executing jobs. To view the list of job templates, navigate to **Hosts > Job templates**. If want to use a template without making changes, proceed to [Section 5.3.2, “Executing Jobs”](#).

You can use default templates as a base for developing your own. Default job templates are locked for editing. Clone the template and edit the clone.

1. To clone a template, in the **Actions** column, select **Clone**.
2. Enter a unique name for the clone and click **Submit** to save the changes.

Job templates use the Embedded Ruby (ERB) syntax. For more information about writing templates, see the [Appendix A, *Template Writing Reference*](#).

Ansible Considerations

To create an Ansible job template, use the following procedure and instead of ERB syntax, use YAML syntax. Begin the template with `---` and to the first line, you must add `hosts: all`. You can embed an Ansible playbook YAML file into the job template body. You can also add ERB syntax to customize your YAML Ansible template. You can also import Ansible playbooks in Satellite. For more information, see [Synchronizing Templates with Git](#) in the *Content Management Guide*.

Parameter Variables

At run time, job templates can accept parameter variables that you define for a host. Note that only the parameters visible on the **Parameters** tab at the host’s edit page can be used as input parameters for job templates. If you do not want your Ansible job template to accept parameter variables at run time, in the Satellite web UI, navigate to **Administer > Settings** and click the **Ansible** tab. In the **Top level Ansible variables** row, change the **Value** parameter to **No**.

To Create a Job Template:

1. Navigate to **Hosts > Job templates**.
2. Click **New Job Template**.
3. Click the **Template** tab, and in the **Name** field, enter a unique name for your job template.
4. Select **Default** to make the template available for all organizations and locations.
5. Create the template directly in the template editor or upload it from a text file by clicking **Import**.
6. Optional: In the **Audit Comment** field, add information about the change.
7. Click the **Job** tab, and in the **Job category** field, enter your own category or select from the default categories listed in [Table 5.1, “Default Job Template Categories”](#).
8. Optional: In the **Description Format** field, enter a description template. For example, **Install package `{package_name}`**. You can also use `{template_name}` and `{job_category}` in your template.

9. From the **Provider Type** menu, select **SSH** for shell scripts and **Ansible** for Ansible tasks or playbooks.
10. Optional: In the **Timeout to kill** field, enter a timeout value to terminate the job if it does not complete.
11. Optional: Click **Add Input** to define an input parameter. Parameters are requested when executing the job and do not have to be defined in the template. For examples, see the **Help** tab.
12. Optional: Click **Foreign input set** to include other templates in this job.
13. Optional: In the **Effective user** area, configure a user if the command cannot use the default `remote_execution_effective_user` setting.
14. Optional: If this template is a snippet to be included in other templates, click the **Type** tab and select **Snippet**.
15. Click the **Location** tab and add the locations where you want to use the template.
16. Click the **Organizations** tab and add the organizations where you want to use the template.
17. Click **Submit** to save your changes.

You can create advanced templates by including other templates in the template syntax, see [Section 5.3.4, “Creating Advanced Templates”](#) for more information. An advanced template is required, for example, for executing jobs that perform power actions; see [Example 5.4, “Including Power Actions in Templates”](#) for information on how to include the **Power Action - SSH Default** template in a custom template.

Table 5.1. Default Job Template Categories

Job template category	Description
Packages	Templates for performing package related actions. Install, update, and remove actions are included by default.
Puppet	Templates for executing Puppet runs on target hosts.
Power	Templates for performing power related actions. Restart and shutdown actions are included by default.
Commands	Templates for executing custom commands on remote hosts.
Services	Templates for performing service related actions. Start, stop, restart, and status actions are included by default.
Katello	Templates for performing content related actions. These templates are used mainly from different parts of the Satellite web UI (for example bulk actions UI for content hosts), but can be used separately to perform operations such as errata installation.

Example 5.1. Creating a restorecon Template

This example shows how to create a template called **Run Command - restorecon** that will restore the default **SELinux** context for all files in the selected directory on target hosts.

1. Navigate to **Hosts > Job templates**. Click **New Job Template**.
2. Enter **Run Command - restorecon** in the **Name** field. Select **Default** to make the template available to all organizations. Add the following text to the template editor:

```
restorecon -RvF <%= input("directory") %>
```

The `<%= input("directory") %>` string will be replaced by a user-defined directory during job invocation.

3. On the **Job** tab, set **Job category** to **Commands**.
4. Click **Add Input** to allow job customization. Enter **directory** to the **Name** field. The input name must match the value specified in the template editor.
5. Click **Required** so that the command cannot be executed without the user specified parameter.
6. Select **User input** from the **Input type** drop-down list. Enter a description to be shown during job invocation, for example **Target directory for restorecon**.
7. Click **Submit**.

See [Example 5.2, “Executing a restorecon Template on Multiple Hosts”](#) for information on how to execute a job based on this template.

5.3.2. Executing Jobs

This section shows how to run a job based on a job template against one or more hosts.

To Execute a Remote Job:

1. Navigate to **Hosts > All hosts** and select the target hosts for your job. You can use the search field to filter the host list.
2. From the **Select Action** menu, select **Schedule Remote Job**.
3. On the **Job invocation** page, define the main job settings:
 - a. Select the **Job category** and the job template you want to use.
 - b. Optionally, select a stored search string in the **Bookmark** list to specify the target hosts.
 - c. Optionally, further limit the targeted hosts by entering a **Search query**. The **Resolves to** line displays the number of hosts affected by your query. Use the refresh button to recalculate the number after changing the query. The preview icon will list the targeted hosts.
 - d. The remaining settings depend on the selected job template. See [To Create a Job Template](#): for information on adding custom parameters to a template.

- Clicking **Display advanced fields** will show advanced setting for the job. Some of the advanced settings depend on the job template, the following settings are general:
 - Effective user** defines the user for executing the job, by default it is the SSH user.
 - Concurrency level** defines maximum number of jobs executed at once, which can prevent overload of systems' resources in a case of executing the job on a large number of hosts.
 - Time span** defines time interval in seconds after which the job should be killed, if it is not finished already. A task which could not be started during the defined interval, for example, if the previous task took too long to finish, is canceled.
 - Type of query** defines when the search query is evaluated. This helps to keep the query up to date for scheduled tasks.
Concurrency level and **Time span** settings enable you to tailor job execution to fit your infrastructure hardware and needs.
- If you want to run the job immediately, ensure that **Schedule** is set to **Execute now**. You can also define a one-time future job, or set up a recurring job. For recurring tasks, you can define start and end dates, number and frequency of runs. You can also use cron syntax to define repetition. For more information about cron, see the [Automating System Tasks](#) section of the Red Hat Enterprise Linux 7 *System Administrator's Guide*.
- Click **Submit**. This displays the **Job Overview** page, and when the job completes, also displays the status of the job.

Example 5.2. Executing a restorecon Template on Multiple Hosts

This example shows how to run a job based on the template created in [Example 5.1, "Creating a restorecon Template"](#) on multiple hosts. The job will restore the SELinux context in all files under the **/home/** directory.

- Navigate to **Hosts > All hosts** and select target hosts. Select **Schedule Remote Job** from the **Select Action** drop-down list.
- In the **Job invocation** page, select the **Commands** job category and the **Run Command - restorecon** job template.
- Type **/home** in the **directory** field.
- Set **Schedule** to **Execute now**.
- Click **Submit**. You are taken to the **Job invocation** page where you can monitor the status of job execution.

5.3.3. Monitoring Jobs

You can monitor the progress of the job while it is running. This can help in any troubleshooting that may be required.

To Monitor a Job:

- Navigate to the Job page. This page is automatically displayed if you triggered the job with the **Execute now** setting. To monitor scheduled jobs, navigate to **Monitor > Jobs** and select the job run you wish to inspect.

2. On the Job page, click the **Hosts** tab. This displays the list of hosts on which the job is running.
3. In the **Host** column, click the name of the host that you want to inspect. This displays the **Detail of Commands** page where you can monitor the job execution in real time.
4. Click **Back to Job** at any time to return to the **Job Details** page.

5.3.4. Creating Advanced Templates

When creating a job template, you can include an existing template in the template editor field. This way you can combine templates, or create more specific templates from the general ones.

The following template combines default templates to install and start the **httpd** service on Red Hat Enterprise Linux systems:

```
<%= render_template 'Package Action - SSH Default', :action => 'install',
:package => 'httpd' %>
<%= render_template 'Service Action - SSH Default', :action => 'start',
:service_name => 'httpd' %>
```

The above template specifies parameter values for the rendered template directly. It is also possible to use the **input()** method to allow users to define input for the rendered template on job execution. For example, you can use the following syntax:

```
<%= render_template 'Package Action - SSH Default', :action => 'install',
:package => input("package") %>
```

With the above template, you have to import the parameter definition from the rendered template. To do so, navigate to the **Jobs** tab, click **Add Foreign Input Set**, and select the rendered template from the **Target template** drop-down list. You can import all parameters or specify a comma separated list.

Example 5.3. Rendering a restorecon Template

This example shows how to create a template derived from the **Run command - restorecon** template created in [Example 5.1, “Creating a restorecon Template”](#). This template does not require user input on job execution, it will restore the SELinux context in all files under the **/home/** directory on target hosts.

Create a new template as described in [Section 5.3.1, “Setting up Job Templates”](#), and specify the following string in the template editor:

```
<%= render_template("Run Command - restorecon", :directory => "/home")
%>
```

Example 5.4. Including Power Actions in Templates

This example shows how to set up a job template for performing power actions, such as reboot. This procedure prevents Satellite from interpreting the disconnect exception upon reboot as an error, and consequently, remote execution of the job works correctly.

Create a new template as described in [Section 5.3.1, “Setting up Job Templates”](#), and specify the following string in the template editor:

```
<%= render_template("Power Action - SSH Default", :action => "restart")
%>
```

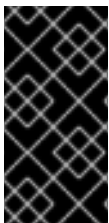
5.4. CONFIGURING GLOBAL SETTINGS

The Satellite remote execution feature provides numerous global settings that you can use to configure its behavior. These are listed in [Table 5.2, “Global Settings for Remote Execution”](#). To review and update these settings, navigate to **Administer > Settings** and click the **Remote Execution** tab.

Table 5.2. Global Settings for Remote Execution

Parameter Name	Description
Default SSH key passphrase	Defines the default key passphrase to use for SSH. You can override this for each host by setting the remote_execution_ssh_key_passphrase parameter.
Enable Global Capsule	When enabled, searches the host for any Capsule with remote execution configured. This is useful when the host has no subnet configured or if the subnet does not have a Capsule with remote execution enabled.
Fallback Without Capsule	When enabled, the remote execution will try to run the commands directly, when no Capsule with remote execution feature is configured for the host.
Fallback to Any Capsule	When enabled, searches for a remote execution Capsule outside of the Capsules assigned to the host. If the location or organization is configured, the search will be limited to the host’s organization or location.
Effective User Method	Defines which method to use to set the effective user on the target host. One of sudo , dzdo , su .
Effective User	Defines the default effective user for any job. When the job is executed, the effective user of the process is changed accordingly. You can override this for each job template and job invocation.
Connect by IP	When enabled, the IP addresses on host interfaces is preferred over the FQDN. It is useful when DNS is not resolving the FQDNs properly. You can override this for each host by setting the remote_execution_connect_by_ip parameter.
SSH User	Defines the default user to use while the Capsule connects to the target using SSH. You can override this for each host by setting the remote_execution_ssh_user parameter. You can set this by host, host group, operating system, domain, location, or organization. This can also be a different user from the effective user.
Sudo password	Defines the sudo password.

Parameter Name	Description
Default SSH password	Defines the default password to use for SSH. You can override this for each host by setting the remote_execution_ssh_password parameter.
SSH Port	Defines the port to use for SSH communication. The default port is 22. You can override this for each host by setting the remote_execution_ssh_port parameter.
Sync Job Templates	When enabled, job templates are synchronized from disk when seeding a database.
Cleanup working directories	When enabled, working directories will be removed after task completion. You can override this for each host by setting a remote_execution_cleanup_working_dirs parameter.
Workers pool size	Defines the number of workers in the pool to handle the execution of the remote execution jobs. Restart of the dynflowd/foreman-tasks service is required.



IMPORTANT

Modify these parameters in the web UI, because any manual changes that you make in the `/etc/foreman/settings.yaml` file are overwritten the next time you run **satellite-installer**. Alternatively, use the **foreman-rake config** command from a console.

5.4.1. Choosing a Capsule for Remote Execution

Remote execution requires a Capsule Server to perform any specified job on a host. By default, any Capsule within the host's organization and location with the **remote execution provider** feature enabled is considered available to perform these jobs. You can set the **remote_execution_global_proxy** variable to **false** to disable this behavior. This may be necessary in more complex environments, where not all Capsules can be used due to possible network isolation. In this configuration, you can assign a pool of Capsules to each subnet, and jobs are load balanced across them.

Alternatively, you can set the **remote_execution_fallback_proxy** variable to **true** to enable fallback mode. In this configuration, remote execution will use any Capsule associated with the host, such as its Puppet Master, provided that Capsule also has remote execution configured.

5.5. DELEGATING PERMISSIONS FOR REMOTE EXECUTION

You can control which users can run which jobs within your infrastructure, including which hosts they can target. The remote execution feature provides two built-in roles:

- **Remote Execution Manager:** This role allows access to all remote execution features and functionality.
- **Remote Execution User:** This role only allows running jobs; it does not provide permission to modify job templates.

You can clone the Remote Execution User role and customize its filter for increased granularity. If you adjust the filter with the **view_job_templates** permission, the user can only see and trigger jobs based on matching job templates. You can use the **view_hosts** and **view_smart_proxies** permissions to limit which hosts or Capsules are visible to the role.

The **execute_template_invocation** permission is a special permission that is checked immediately before execution of a job begins. This permission defines which job template you can run on a particular host. This allows for even more granularity when specifying permissions. For more information on working with roles and permissions see [Creating and Managing Roles](#) in the *Administering Red Hat Satellite*.

The following example shows filters for the **execute_template_invocation** permission:

```
name = Reboot and host.name = staging.example.com
name = Reboot and host.name ~ *.staging.example.com
name = "Restart service" and host_group.name = webservers
```

The first line in the above example permits the user to apply the **Reboot** template to one selected host. The second line defines a pool of hosts with names ending with **.staging.example.com**. The third line binds the template with a host group.



NOTE

Permissions assigned to users can change over time. If a user has already scheduled some jobs to run in the future, and the permissions have changed, this can result in execution failure because the permissions are checked immediately before job execution.

CHAPTER 6. DISCOVERING BARE-METAL HOSTS ON SATELLITE

Red Hat Satellite 6.5-beta includes the Discovery plug-in. The Discovery plug-in enables automatic bare-metal discovery of unknown hosts on the provisioning network. These new hosts are registered to the Satellite Server and the Puppet agent on the client uploads system facts collected by Facter, such as serial ID, network interface, memory, and disk information. After registration, the hosts are displayed on the **Discovered Hosts** page in the Satellite web UI. You can then initiate provisioning either manually (using the web UI, CLI, or API) or automatically, using predefined discovery rules.

The Discovery plug-in communicates through the Satellite Capsule Server, which has direct access both to the provisioning network and the Satellite Server instance. It is possible to discover hosts directly from the Satellite Server, but Red Hat recommends the following scheme be used:

```
Satellite Server (Satellite Server Discovery plug-in) <--> Satellite
Capsule (Satellite Capsule Discovery plug-in) <--> Discovered Host
(Satellite Discovery image)
```

The Satellite Discovery plug-in consists of three different components:

The Satellite Server Discovery plug-in

This runs on the Satellite Server and provides API and UI functionality for working with discovered hosts. The `tfm-rubygem-foreman_discovery` package contains this plug-in.

The Satellite Capsule Server Discovery plug-in

This is a communication proxy between discovered hosts on a provisioning network and the Satellite Server. The `rubygem-smart_proxy_discovery` package contains this plug-in.

The Satellite Discovery image

This is the minimal operating system based on Red Hat Enterprise Linux that is PXE-booted on hosts to acquire initial hardware information and to check in to the Satellite Server. Discovered hosts keep running the Satellite Discovery image until they are rebooted into Anaconda, which then initiates the provisioning process. The `foreman-discovery-image` package contains this image. It must be installed on the Satellite Capsule Server that provides TFTP services.

6.1. NETWORK CONFIGURATION FOR PXE-BASED DISCOVERY

The discovery process is based on PXE: Systems must boot from the network using a single Ethernet connection to the LAN or VLAN. All other network interface configurations are not supported (bonding, teaming, bridging, DSL, Wi-Fi and others).

You must have a separate LAN or VLAN for discovery and PXE provisioning. You can configure systems to use VLAN trunks, but you must also configure the provisioning interface with the correct VLAN tag for the provisioning VLAN, and then change the tag to the production VLAN using a post-installation script.

Although using special network configurations is technically possible in PXE-less mode, where discovered systems use kexec to load a new kernel with Anaconda which avoids PXE booting completely, the discovery image currently does not allow such a configuration. While it is possible to use discovery extensions or a script to re-configure the network, Satellite 6 discovery plug-in cannot work with such a configuration.

Because the discovery process currently has limited possibilities for configuring network interfaces, and because the provisioning interface is also the primary interface, to simplify the configuration, have a

separate primary interface from the interface used in production. Satellite 6 template features can be used to deploy post-installation scripts to configure interfaces if required.

6.2. CONFIGURING THE SATELLITE DISCOVERY PLUG-IN

The following sections describe how to configure the Satellite Discovery plug-in and how to prepare the PXE-boot template on the Satellite Server.

6.2.1. Deploying the Satellite Discovery Image

Install the package containing the Satellite Discovery image on the Satellite Capsule Server that provides TFTP services (not on the Satellite Server itself):

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

This package contains the Linux kernel and initial RAM disk image as a bootable ISO file which is used for PXE-booting discovered hosts. You can run the following command to investigate the contents of the package. This produces output similar to the following:

```
$ rpm -ql foreman-discovery-image
/usr/share/foreman-discovery-image
/usr/share/foreman-discovery-image/fdi-image-rhel_7-2.1.0-20150212.1.iso
```

When you install this package, it extracts the kernel and image from the ISO file into the TFTP directory and creates symbolic links to the latest versions of the image and kernel. Use the symbolic links in the PXE-boot provisioning template to make sure that you do not need to change the version in the template every time the **foreman-discovery-image** package is upgraded. For example:

```
$ find /var/lib/tftpboot/boot
/var/lib/tftpboot/boot
/var/lib/tftpboot/boot/fdi-image-rhel_7-2.1.0-20150212.1-img
/var/lib/tftpboot/boot/fdi-image-rhel_7-2.1.0-20150212.1-vmlinuz
/var/lib/tftpboot/boot/fdi-image-rhel_7-img
/var/lib/tftpboot/boot/fdi-image-rhel_7-vmlinuz
```



NOTE

Currently, only Red Hat Enterprise Linux 7 Discovery images are provided, even for Satellite 6 installations on Red Hat Enterprise Linux 6. If there are discovered hosts running during the upgrade of the **foreman-discovery-image** package, reboot them all to load the updated version of the image as soon as possible. This can be done using the Satellite 6 web UI, CLI, or API.

6.2.2. Configuring PXE-booting

When an unknown host is booted on the provisioning network, Satellite Server provides a PXELinux boot menu with a single option: to boot from the local hard drive. You can use following procedure to build a default PXE template in Satellite to enable hardware discovery.

To Configure PXE-booting for host discovery:

1. In the Satellite web UI, navigate to **Hosts > Provisioning Templates**.

2. In the upper-right of the **Provisioning Templates** page, click **Build PXE Default**, and click **OK**.

The template becomes the default template on all TFTP servers. Every new unknown host that is in the provisioning subnet uses this configuration and uses the Foreman Discovery Image as the default.

6.2.3. Reviewing Global Discovery Settings

You can review global settings related to the Discovery plug-in in the Satellite web UI. Navigate to **Administer > Settings** and open the **Discovered** tab. Notable settings are:

Discovery organization, Discovery location

These variables specify where to place the discovered hosts. By default, the discovered hosts are automatically placed under the first organization and location created.

Interface fact

This variable specifies which incoming fact to use to determine the MAC address of the discovered host. By default, the PXELinux BOOTIF kernel command line option is used.

Hostname facts

This variable allows you to list facts to use for the host name. These are separated by commas, and the first fact in the list takes precedence.

Auto provisioning

This variable enables automatic provisioning according to specified rules. Set to false by default. Red Hat recommends that you test the configuration with manual provisioning before enabling Auto provisioning. See [Section 6.4, “Provisioning Discovered Hosts”](#) for more information.

Reboot

This variable enables automatic reboot of a host discovered by PXE, or the use of kexec for a host booted from local media, during provisioning. This is set to true by default.

Hostname prefix

This variable specifies the default prefix to use for the host name. Set to "mac" by default. The variable must start with a letter.

Fact columns

This variable allows you to add any fact reported by Facter as an additional column in discovered host lists.

Highlighted facts

This variable uses regular expressions to organize facts for the highlights section.

Storage facts

This variable uses regular expressions to organize facts for the storage section.

Hardware facts

This variable uses regular expressions to organize facts for the hardware section.

Network facts

This variable uses regular expressions to organize facts for the network section.

IPMI facts

This variable uses regular expressions to organize facts for the IPMI section.

6.3. CONFIGURING THE SATELLITE CAPSULE SERVER DISCOVERY PLUG-IN

Ensure the `foreman_url` setting exists in the Satellite Capsule Server configuration file. The setting can appear as follows:

```
# grep foreman_url /etc/foreman-proxy/settings.yml
:foreman_url: https://satellite.example.com
```

The `satellite-installer` command configures this variable automatically, but Red Hat recommends that you check that the host responds correctly and there are no firewall rules blocking communication.

6.3.1. Configuring Discovery Subnets

You need to configure all subnets with discovered hosts to communicate to the Satellite Server or a Capsule Server for host discovery and provisioning. You must first enable a Capsule to provide a proxy service for provisioning templates for the subnet. The template Capsule for a subnet does not have to be the same Capsule that is set as the TFTP Capsule.

If you want to configure a new subnet, see [Adding a Subnet to the Satellite Server](#) in the *Provisioning Guide*.

To enable a Template Capsule:

You can enable the Satellite's integrated Capsule and any external Capsule Servers to provide a proxy service for provisioning templates with the following command:

```
# satellite-installer --foreman-proxy-templates=1
```

To verify that a Capsule Server provides a template proxy service:

1. Navigate to **Infrastructure > Capsules**.
2. From the **Actions** menu, select **Refresh** to ensure that the list is up-to-date.
3. In the **Features** column, search for the word "Templates".

To Configure a Subnet with a TFTP Capsule, Template Capsule, and Discovery Capsule:

1. In the Satellite web UI, navigate to **Infrastructure > Subnets**.
2. Select the subnet you want to configure.
3. On the **Capsules** tab, select the **TFTP Capsule**, **Template Capsule**, and **Discovery Capsule** for this subnet.

6.3.2. Using Hammer with the Discovery Plug-in

To use the `hammer` command with the Discovery plug-in, you need to enable the Discovery plug-in in `/etc/hammer/cli.modules.d/foreman_discovery.yml` as follows:

■

```

:foreman_discovery:
  :enable_module: true

```

See [hammer configuration directories](#) for more information about the files and directories that **hammer** uses.

6.3.3. Reviewing User Permissions

When it first starts, the Satellite Capsule Server Discovery plug-in creates a role called **Discovery**. You can assign this role to non-administrative users to allow them to use the Discovery plug-in. Alternatively, assign the **perform_discovery** permission to an existing role. For more information on roles and permissions, see [Creating and Managing Users](#) in *Administering Red Hat Satellite*.

6.4. PROVISIONING DISCOVERED HOSTS

After you have correctly configured Discovery plug-ins on both Satellite Server and Capsule Server, you can automatically detect bare-metal hosts. To do so, boot a machine in any provisioning network that was configured with the PXE configuration template described in [Section 6.2.2, “Configuring PXE-booting”](#). The machine is automatically registered with the Satellite Server and appears in the **Hosts > Discovered Hosts** list in the Satellite web UI.

You can either provision the discovered host manually, or you can configure automatic provisioning.

6.4.1. Manually Provisioning Hosts

The following procedure describes how to manually provision discovered hosts from the Satellite web UI.

To Manually Provision a Discovered Host:

1. Navigate to **Hosts > Discovered Hosts**.
2. Select the host you want to provision and click **Provision**.
3. On the host’s **Edit** page, complete the necessary details, and then click **Save**.

When the host configuration is saved, Satellite modifies the host’s PXELinux file on the TFTP server and reboots the discovered host. It then boots into an installer for the chosen operating system, and finally into the installed operating system.

If you decide to re-provision an existing discovered host, delete the operating system from the machine and reboot it. The host then reappears on the **Discovered Hosts** page.

6.4.2. Decommissioning Discovered Hosts

If you no longer require Red Hat Satellite to manage a specific host, you need to decommission that host to prevent it from being discovered.

To Decommission a Discovered Host:

1. Shut down the host.
2. Navigate to **Hosts > Discovered Hosts**.

3. In the **Name** column find the host you want to decommission and then select **Delete** from the corresponding **Edit** drop-down menu.

6.4.3. Automatically Provisioning Hosts

With Satellite 6.5-beta, it is possible to define provisioning rules that will assign a host group to provisioned hosts and trigger provisioning automatically.

To Create a Provisioning Rule:

1. Navigate to **Configure > Discovery rules**.
2. Click **New Rule**. Specify the following parameters of the provisioning rule:
 - **Name** is the name of the rule displayed in the list of rules. This name must not contain spaces or non-alphanumeric characters.
 - **Search** is the search statement used to match discovered hosts for the particular rule. You can use scoped search syntax to define it. See [Section 6.4.4, "Scoped Search Syntax"](#) for examples of using scoped search.
 - **Host Group** is the host group to be assigned to a matching host before starting the provisioning process. Make sure that the selected host group has all the required parameters set; required parameters are marked with an asterisk (*).
 - **Hostname** defines a pattern for assigning human-readable host names to the matching hosts. When left blank, the host name is assigned in the format "macMACADDRESS" by default. The same syntax used for provisioning templates is used in this instance. See [Section 6.4.5, "Host Name Patterns"](#) for more information and examples.
 - **Hosts limit** is the maximum number of provisioned hosts per rule. If the limit is reached, the rule will not take effect until one or more hosts are deleted. Typical use cases are rules per server rack or row when it is necessary to change provisioning parameters such as host name or host group per entry. You can set this value to zero (0) to specify no limit.
 - **Priority** specifies the order of execution of rules. The value must be greater than or equal to zero. A lower value indicates a higher priority. If two rules have the same priority, the first rule encountered is applied.
 - **Enabled** provides the option to temporarily enable or disable rules.
3. Click **Submit** to save the rule.

By default, Satellite does not enable automatic discovery of hosts. The following procedure describes how to enable the Auto provisioning variable to provide automatic provisioning according to specified rules.

To Enable Automatic Provisioning:

1. Navigate to **Administer > Settings > Discovered** in the Satellite web UI.
2. Locate Auto provisioning in the **Name** column, and set its value to **true**.
3. Click **Save**.

After you have defined some rules, Red Hat recommends that you discover a host and apply the rules using the **Auto discover** button on the host. This triggers auto-provisioning without the need to enable the global option.

6.4.4. Scoped Search Syntax

This section shows how to use scoped search syntax to filter the discovered hosts according to selected parameters. This is useful when creating a rule for automatic provisioning (see [Section 6.4.3, “Automatically Provisioning Hosts”](#)).

The search fields in the Satellite web UI support automatic completion to make building search strings easier. For example, you can test search patterns on the **Hosts > Discovered Hosts** page. The following are examples of typical search queries:

- `facts.architecture = x86_64`
- `facts.bios_vendor ~ 'Dell*'`
- `facts.macaddress = "aa:bb:cc:dd:ee:ff"`
- `facts.macaddress_eth0 = "aa:bb:cc:dd:ee:ff"`
- `facts.ipaddress_eth1 ~ "192.168.*"`
- `facts.architecture ^ (x86_64,i386)`



NOTE

The caret symbol (^) in scoped searches means "in" (the same usage as in SQL) and not "starts with" as it is used in regular expressions. You can review the full list of scoped search operators at

https://github.com/wvanbergen/scoped_search/blob/master/lib/scoped_search/query_language

In Satellite 6.5-beta, all facts are strings, so it is not possible to do numeric comparisons. However, three important facts are extracted and converted to numbers. These are described in [Table 6.1, “Facts that Allow Numerical Comparison”](#).

Table 6.1. Facts that Allow Numerical Comparison

Search Parameter	Description	Example Usage
cpu_count	The number of CPUs	<code>cpu_count >= 8</code>
disk_count	The number of disks attached	<code>disk_count < 10</code>
disks_size	The total amount of disk space (in MiB)	<code>disks_size > 1000000</code>

6.4.5. Host Name Patterns

This section lists the host name patterns that you can use when creating a rule for automatic provisioning (see [Section 6.4.3, “Automatically Provisioning Hosts”](#)).

The target host name template pattern has the same syntax as the provisioning templates (ERB). The domain is appended automatically. In addition to the **@host** attribute, the **rand()** function for random integers is available. For example:

- application-server-<%= rand(99999) %>
- load-balancer-<%= @host.facts['bios_vendor'] + '-' + rand(99999) %>
- wwwsrv-<%= @host.hostgroup.name %>
- minion-<%= @host.discovery_rule.name %>
- db-server-<%= @host.ip.gsub('.', '-') + '-' + @host.hostgroup.subnet.name %>



IMPORTANT

When creating host name patterns, ensure the resulting host names are unique. Host names must not start with numbers. A good approach is to use unique information provided by Factor (for example, the MAC address, BIOS or serial ID) or to otherwise randomize the host name.

6.4.6. Using the Discovery Plug-in on the Command Line

You can use the **hammer** command to perform certain tasks related to discovery. Run the **hammer -h** command to verify your configuration:

```
$ hammer -h | grep discovery
discovery                Manipulate discovered hosts.
discovery_rule           Manipulate discovered rules.
```

Use the **hammer discovery -h** command to view the available options. For example, you can use the following command to reboot a discovered host (assuming its ID is 130):

```
$ hammer discovery reboot -id 130
Host reboot started
```

6.5. EXTENDING THE DISCOVERY IMAGE

It is possible to extend the Satellite Discovery image with custom facts, software, or device drivers. You can also provide a compressed archive file containing extra code for the image to use.

First, create the following directory structure:

```
.
├── autostart.d
│   └── 01_zip.sh
├── bin
│   └── ntpdate
├── facts
│   └── test.rb
├── lib
│   ├── libcrypto.so.1.0.0
│   └── ruby
│       └── test.rb
```

Where:

- The **autostart.d** directory contains scripts that are executed in POSIX order by the image when it starts, but before the host is registered to Satellite.
- The **bin** directory is added to the `$PATH` variable; you can place binary files here and use them in the autostart scripts.
- The **facts** directory is added to the `FACTERLIB` variable so that custom facts can be configured and sent to Satellite.
- The **lib** directory is added to the `LD_LIBRARY_PATH` variable and **lib/ruby** is added to the `RUBYLIB` variable, so that binary files in `/bin` can be executed correctly.

New directives and options are appended to the existing environment variables (`PATH`, `LD_LIBRARY_PATH`, `RUBYLIB` and `FACTERLIB`). If you need to specify the path to something explicitly in your scripts, the zip contents are extracted to the `/opt/extension` directory on the image.

After creating the above directory structure, package it into a zip archive with the following command:

```
zip -r my_extension.zip .
```

You can create multiple zip files but be aware they will be extracted to the same place on the Discovery image, so files in later zips will overwrite earlier ones if they have the same file name.

To inform the Discovery image of the extensions it should use, place your zip files on your TFTP server with the Discovery image, and then update the `APPEND` line of the PXELinux template with the **fdi.zips** option where the paths are relative to the TFTP root. For example, if you have two archives at `$TFTP/zip1.zip` and `$TFTP/boot/zip2.zip`, use the following syntax:

```
fdi.zips=zip1.zip,boot/zip2.zip
```

See [Section 6.2.2, “Configuring PXE-booting”](#) for more information on updating the PXE template.

6.6. TROUBLESHOOTING SATELLITE DISCOVERY

If a machine is not listed in the Satellite web UI under **Hosts > Discovered Hosts**, inspect the following configuration areas to help isolate the error:

- Navigate to **Hosts > Provisioning Templates** and redeploy the default PXELinux template using the **Build PXE Default** button.
- Verify the `pxelinux.cfg/default` configuration file on the TFTP Capsule Server.
- Ensure adequate network connectivity between hosts, Capsule Server, and Satellite Server.
- Check the PXELinux template in use and determine the PXE discovery snippet it includes. Snippets are named as follows: **pxelinux_discovery**, **pxegrub_discovery**, or **pxegrub2_discovery**. Verify the `proxy.url` and `proxy.type` options in the PXE discovery snippet.
- Ensure that DNS is working correctly for the discovered nodes, or use an IP address in the `proxy.url` option in the PXE discovery snippet included in the PXELinux template you are using.

- Ensure that the DHCP server is delivering IP addresses to the booted image correctly.
- Ensure the discovered host or virtual machine has at least 1200 MB of memory. Less memory can lead to various random kernel panic errors as the image needs to be extracted in-memory.

For gathering important system facts, use the **discovery-debug** command. It prints out system logs, network configuration, list of facts, and other information on the standard output. The typical use case is to redirect this output and copy it with the **scp** command for further investigation.

The first virtual console on the discovered host is reserved for **systemd** logs. Particularly useful system logs are tagged as follows:

- `discover-host` — initial facts upload
- `foreman-discovery` — facts refresh, reboot remote commands
- `nm-prepare` — boot script which pre-configures NetworkManager
- `NetworkManager` — networking information

Use TTY2 or higher to log in to a discovered host. The root account and SSH access are disabled by default, but you can enable SSH and set the root password using the following kernel command-line options in the Default PXELinux template on the APPEND line:

```
fdi.ssh=1 fdi.rootpw=redhat
```

CHAPTER 7. INTEGRATING RED HAT SATELLITE AND ANSIBLE TOWER

You can integrate Red Hat Satellite 6.5-beta and Ansible Tower to use Satellite Server as a dynamic inventory source for Ansible Tower.

You can also use the provisioning callback function to run playbooks on hosts managed by Satellite, from either the host or Ansible Tower. When provisioning new hosts from Satellite Server, you can use the provisioning callback function to trigger playbook runs from Ansible Tower. The playbook configures the host following Kickstart deployment.

7.1. ADDING SATELLITE SERVER TO ANSIBLE TOWER AS A DYNAMIC INVENTORY ITEM

To add Satellite Server to Ansible Tower as a dynamic inventory item, you must create a credential for a Satellite Server user on Ansible Tower, add an Ansible Tower user to the credential, and then configure an inventory source.

Prerequisites

- You must have a Satellite Server user with an integration role that includes the necessary permission filters. For more information about managing users, roles, and permission filters, see [Managing Users and Roles](#) and [Creating and Managing Roles](#) in *Administering Red Hat Satellite*.
- You must specify the following permission filters and assign the role to the user.

Table 7.1. Permission Filters

Resource	Permissions	Access Description
Host	<code>view_hosts</code>	To view Satellite Server hosts.
Host Group	<code>view_hostgroups</code>	To view Satellite Server host groups.
Fact value	<code>view_facts</code>	To view Satellite Server Facts.

- You must host your Satellite Server and Ansible Tower on the same network or subnet.

Procedure

To add Satellite Server to Ansible Tower as a Dynamic Inventory Item, complete the following procedure:

- In the Ansible Tower web UI, create a credential for your Satellite. For more information about creating credentials, see [Add a New Credential](#) and [Red Hat Satellite 6 Credentials](#) in the *Ansible Tower User Guide*.

Table 7.2. Satellite Credentials

Credential Type:	Red Hat Satellite 6
Satellite 6 URL:	<code>https://satellite.example.com</code>

Credential Type:	Red Hat Satellite 6
Username:	The username of the Satellite user with the integration role.
Password:	The password of the Satellite user.

2. Add an Ansible Tower user to the new credential. For more information about adding a user to a credential, see [Getting Started with Credentials](#) in the *Ansible Tower User Guide*.
3. Add a new inventory. For more information, see [Add a new inventory](#) in the *Ansible Tower User Guide*.
4. In the new inventory, add Satellite Server as the inventory source, specifying the following inventory source options. For more information, see [Add Source](#) in the *Ansible Tower User Guide*.

Table 7.3. Inventory Source Options

Source	Red Hat Satellite 6
Credential	The credential you create for Satellite Server.
Overwrite	Select
Overwrite Variables	Select
Update on Launch	Select
Cache Timeout	90

5. Ensure that you synchronize the source that you add.

7.2. CONFIGURING PROVISIONING CALLBACK FOR A HOST

When you create hosts in Satellite, you can use Ansible Tower to run playbooks to configure your newly created hosts. This is called *provisioning callback* in Ansible Tower.

The provisioning callback function triggers a playbook run from Ansible Tower as part of the provisioning process. The playbook configures the host after Kickstart deployment.

For more information about provisioning callbacks, see [Provisioning Callbacks](#) in the *Ansible Tower User Guide*.

In Satellite Server, the **Kickstart Default** and **Kickstart Default Finish** templates include three snippets:

1. `ansible_provisioning_callback`
2. `ansible_tower_callback_script`

3. `ansible_tower_callback_service`

You can add parameters to hosts or host groups to provide the credentials that these snippets can use to run Ansible playbooks on your newly created hosts.

Prerequisites

Before you can configure provisioning callbacks, you must add Satellite as a dynamic inventory in Ansible Tower. For more information, see [Integrating Satellite and Ansible Tower](#).

In the Ansible Tower web UI, you must complete the following tasks:

1. Create a machine credential for your new host. Ensure that you enter the same password in the credential that you plan to assign to the host that you create in Satellite. For more information, see [Add a New Credential](#) in the *Ansible Tower User Guide*.
2. Create a project. For more information, see [Projects](#) in the *Ansible Tower User Guide*.
3. Add a job template to your project. For more information, see [Job Templates](#) in the *Ansible Tower User Guide*.
4. In your job template, you must enable provisioning callbacks, generate the host configuration key, and note the `template_ID` of your job template. For more information about job templates, see [Job Templates](#) in the *Ansible Tower User Guide*.

Procedure

To configure provisioning callback for a new host in Satellite, complete the following steps:

1. In the Red Hat Satellite web UI, navigate to **Configure > Host Group**.
2. Create a host group or edit an existing host group.
3. In the Host Group window, click the **Parameters** tab.
4. Click **Add Parameter**.
5. Enter the following information for each new parameter:

Table 7.4. Host Parameters

Name	Value	Description
<code>ansible_tower_provisioning</code>	true	Enables Provisioning Callback.
<code>ansible_tower_fqdn</code>	<code>tower.example.com</code>	The fully qualified domain name (FQDN) of your Ansible Tower. Do not add https because this is appended by Ansible Tower.
<code>ansible_job_template_id</code>	<code>template_ID</code>	The ID of your provisioning template that you can find in the URL of the template: /templates/job_template/5.
<code>ansible_host_config_key</code>	<code>config_KEY</code>	The host configuration key that your job template generates in Ansible Tower.

6. Click **Submit**.
7. Create a host using the host group.
8. On the new host, enter the following command to start the **ansible-callback** service:

```
# systemctl start ansible-callback
```

9. On the new host, enter the following command to output the status of the **ansible-callback** service:

```
# systemctl status ansible-callback
```

Provisioning callback is configured correctly if the command returns the following output:

```
SAT_host systemd[1]: Started Provisioning callback to Ansible Tower...
```

Manual Provisioning Callback

You can use the provisioning callback URL and the host configuration key from a host to call Ansible Tower. For example:

```
# curl -k -s --data curl --insecure --data host_config_key=my_config_key \
https://tower.example.com/api/v2/job_templates/8/callback/
```

Ensure that you use **https** when you enter the provisioning callback URL.

This triggers the playbook run specified in the template against the host.

CHAPTER 8. SAMPLE SCENARIOS

8.1. SIMPLE SCENARIO

The simple scenario shows how to add a single host, register it, set up, and run a job on it.

8.1.1. Creating the Host

The following procedure provides an example on how to create a host in Red Hat Satellite.

To Create a Host:

1. Click **Hosts** > **Create Host**.
2. On the **Host** tab, enter the required details:
 - a. In the **Name** field, enter the host name, for example *host1.example.com*.
 - b. In the **Organization** field, enter the organization name, for example *MyOrg*.
 - c. In the **Location** field, enter the location name, for example *MyLoc*.
3. Optionally, on the **Puppet Classes** tab, select the Puppet classes you want to include.
4. On the **Interfaces** tab, edit the primary interface:
 - a. Click the **Edit** button in the **Actions** column.
 - b. Select a type from the **Type** drop-down menu, for example *Interface*.
 - c. In the **MAC address** field, enter the MAC address of your host.
 - d. In the **Device Identifier** field, specify the device identifier for this interface, for example *eth0*.
 - e. In the **DNS name** field, specify the DNS name of your host. For the primary interface, this host name is used with the domain name to form the FQDN.
 - f. Select a domain from the **Domain** drop-down menu, for example *satellite.example.com*. This automatically updates the **IPv4 Subnet** and **IPv6 Subnet** lists with a selection of available subnets. Optionally, select the subnets.
 - g. In the **IPv4 address** field, specify the IPv4 address of your host.
 - h. Click **Ok**.
5. On the **Operating System** tab, enter the required details:
 - a. Select an architecture from the **Architecture** drop-down menu, for example *x86_64*.
 - b. Select an operating system from the **Operating system** drop-down menu, for example *RHEL Server 7.4*.
 - c. Select a partition table from the **Partition table** drop-down menu, for example *Kickstart default*.
 - d. In the **Root password** field, enter the root password for your host.

6. Optionally, on the **Parameters** tab, select the parameters you want to supply to the Puppet master to override the default values.
7. Optionally, on the **Additional Information** tab, enter additional information about the host.
8. Click **Submit**.

8.1.2. Registering the Host

After you have created `host1.example.com`, you must register it so that it can receive updates. The following procedure assumes the host is running Red Hat Enterprise Linux 7.

To Register the Host:

1. In a terminal, connect to the host as the root user.
2. Ensure that a time synchronization tool is enabled and running on the host:

```
# systemctl start chronyd; systemctl enable chronyd
```

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://satellite.example.com/pub/katello-ca-consumer-latest.noarch.rpm
```

4. Ensure that an activation key associated with the appropriate Content View and environment exists for the host. If not, see [Managing Activation Keys](#) in the *Content Management Guide* for more information.
5. Clear any old host data related to Red Hat Subscription Manager (RHSM):

```
# subscription-manager clean
```

6. Register the host using RHSM:

```
# subscription-manager register --org=MyOrg \
--activationkey=my_activation_key
```

Command output after registration:

```
# subscription-manager register --org=MyOrg --
activationkey=my_activation_key
The system has been registered with id: 62edc0f8-855b-4184-b1b8-
72a9dc793b96
```

7. Enable the **Red Hat Satellite Tools 6** repository:

```
# subscription-manager repos --enable=rhel-version-server-satellite-
tools-6-rpms
```

8. Install the **katello-agent**:

-

```
# yum install katello-agent
```

9. Ensure the **goferd** service is running:

```
# systemctl start goferd
```

10. Install and configure the Puppet agent:

- a. Install the Puppet agent:

```
# yum install puppet
```

- b. Configure the Puppet agent to start at boot:

```
# systemctl enable puppet
```

- c. Configure the Puppet agent by specifying the server and environment settings in the `/etc/puppetlabs/puppet/puppet.conf` file:

```
# vi /etc/puppetlabs/puppet/puppet.conf
```

```
[main]
# The Puppet log directory.
# The default value is '$vardir/log'.
logdir = /var/log/puppet

# Where Puppet PID files are kept.
# The default value is '$vardir/run'.
rundir = /var/run/puppet

# Where SSL certificates are kept.
# The default value is '$confdir/ssl'.
ssldir = /var/lib/puppet/ssl

...

[agent]
# The file in which puppetd stores a list of the classes
# associated with the retrieved configuration. Can be
loaded in
# the separate ``puppet`` executable using the ``--
loadclasses``
# option.
# The default value is '$confdir/classes.txt'.
classfile = $vardir/classes.txt
pluginsync = true
report = true
ignoreschedules = true
daemon = false
ca_server = satellite.example.com
server = satellite.example.com
environment = KT_Example_Org_Library_RHEL7Server

# Where puppetd caches the local configuration. An
```

```
# extension indicating the cache format is added
automatically.
# The default value is '$confdir/localconfig'.
localconfig = $vardir/localconfig

...
```

- d. Run the Puppet agent on the host:

```
# puppet agent -t --waitforcert 10 --server satellite.example.com
```

- e. Sign the SSL certificate for the Puppet client in the Satellite Server web UI:

- i. Log in to the Satellite Server web UI.
- ii. Select **Infrastructure > Capsules**.
- iii. Select **Certificates** from the drop-down menu to the right of the required Capsule.
- iv. Click **Sign** to the right of the required host.
- v. Enter the **puppet agent** command again:

```
# puppet agent -t --server satellite.example.com
```

8.1.3. Running a Job on the Host

The following procedure shows how to run a job template on the previously created and registered host *host1.example.com*.

To Execute a Remote Job:

1. Navigate to **Hosts > All hosts** and select the target host, in our example *host1.example.com*.
2. From the **Select Action** menu at the upper right of the screen select **Schedule Remote Job**.
3. On the **Job invocation** page, define the main job settings:
 - a. Select a job category from the **Job category** drop-down menu, for example *Commands*.
 - b. Select a job template from the **Job template** drop-down menu, for example *Run Command - SSH Default*.
 - c. In the **command** field, enter the command you want to run on the host. For example, *timedatectl set-timezone Europe/Prague* will set the time zone to Prague in Europe.
4. Click **Submit**.

APPENDIX A. TEMPLATE WRITING REFERENCE

Embedded Ruby (ERB) is a tool for generating text files based on templates that combine plain text with Ruby code. Red Hat Satellite uses ERB syntax in **provisioning templates** ([Creating Provisioning Templates](#) in the *Provisioning Guide*), remote execution **job templates** ([Chapter 5, Running Jobs on Hosts](#)), **report templates** ([Chapter 3, Generating Host Monitoring Reports](#)), templates for **partition tables** ([Creating Partition Tables](#) in the *Provisioning Guide*), **Smart Variables** ([Configuring Smart Variables](#) in the *Puppet Guide*), and **Smart Class Parameters** ([Configuring Smart Class Parameters](#) in the *Puppet Guide*). This section provides an overview of Satellite specific functions and variables that can be used in ERB templates along with some usage examples. Note that the default templates provided by Red Hat Satellite (**Hosts > Provisioning templates**, **Hosts > Job templates**, **Monitor > Report Templates**) also provide a good source of ERB syntax examples.

When provisioning a host or running a remote job, the code in the ERB is executed and the variables are replaced with the host specific values. This process is referred to as **rendering**. The Satellite Server has the safemode rendering option enabled by default, which prevents any harmful code being executed from templates.

A.1. WRITING ERB TEMPLATES

The following points summarize the ERB syntax:

- `<% %>` – marks enclosing Ruby code within the ERB template. The code is executed when the template is rendered. It can contain Ruby control flow structures as well as Satellite specific functions and variables. For example:

```
<% if @host.operatingsystem.family == "Redhat" &&
@host.operatingsystem.major.to_i > 6 %>
systemctl <%= input("action") %> <%= input("service") %>
<% else %>
service <%= input("service") %> <%= input("action") %>
<% end -%>
```

- `<%= %>` – the code output is inserted into the template. This is useful for variable substitution, for example:

```
echo <%= @host.name %>
```

- `<% -%>`, `<%= -%>` – by default, a newline character is inserted after a Ruby block if it is closed at the end of a line. To suppress this behavior, modify the enclosing mark. For example, the following template:

```
curl <%= @host.ip -%>
/mydir
```

is rendered the same as:

```
curl <%= @host.ip %>/mydir
```

In practice, this is used to reduce the number of lines in rendered templates (where Ruby syntax permits).

- `<%= # %>` – marks enclosing a comment that will be ignored during template rendering:

```
<%# A comment %>
```

A.2. TROUBLESHOOTING ERB TEMPLATES

The Satellite web UI provides two ways to verify the template rendering for a specific host:

- **Directly in the template editor** – when editing a template (under **Hosts** > **Partition tables**, **Hosts** > **Provisioning templates**, or **Hosts** > **Job templates**), on the **Template** tab click **Preview** and select a host from the drop-down menu. The template then renders in the text field using the selected host's parameters. Preview failures can help to identify issues in your template.
- **At the host's details page** – select a host at **Hosts** > **All hosts** and click the **Templates** tab to list templates associated with the host. Select **Review** from the drop-down menu next to the selected template to view it's rendered version.

A.3. SATELLITE SPECIFIC FUNCTIONS AND VARIABLES

This section lists Satellite specific functions and variables for ERB templates. Note that some of them can be used in any kind of template, others are limited, for example job templates accept only @host variables and variables from [Table A.4, "Kickstart Specific Variables"](#) are only applicable in Kickstart templates.

You can use the functions listed in the following table across all kinds of templates.

Table A.1. Generic Functions

Name	Description
indent(n)	Indents the block of code by n spaces, useful when using a snippet template that is not indented.
foreman_url(kind)	Returns the full URL to host-rendered templates of the given kind. For example, templates of the "provision" type usually reside at http://HOST/unattended/provision .
snippet(name)	Renders the specified snippet template. Useful for nesting provisioning templates.
snippets(file)	Renders the specified snippet found in the Foreman database, attempts to load it from the unattended/snippets/ directory if it is not found in the database.
snippet_if_exists(name)	Renders the specified snippet, skips if no snippet with the specified name is found.

Example A.1. Using the snippet and indent Functions

The following syntax imports the **subscription_manager_registration** snippet to the template and indents it by four spaces:

```
<%= indent 4 do
  snippet 'subscription_manager_registration'
end %>
```

The following functions can be used in job templates. See [Section 5.3.4, “Creating Advanced Templates”](#) for usage examples.

Table A.2. Functions Specific to Job Templates

Name	Description
input(input_name)	Returns the value of the specified input on the job execution.
render_template(name, parameters)	Renders the specified template, similar to the generic snippet() function but enables passing arguments to the template.

The following variables enable using host data within templates.

Table A.3. Host Specific Variables and Functions

Name	Description
@host.architecture	The architecture of the host.
@host.bond_interfaces	Returns an array of all bonded interfaces. See Note .
@host.capabilities	The method of system provisioning, can be either build (for example kickstart) or image.
@host.certname	The SSL certificate name of the host.
@host.diskLayout	The disk layout of the host. Can be inherited from the operating system.
@host.domain	The domain of the host.
@host.environment	The Puppet environment of the host.
@host.facts	Returns a Ruby hash of facts from Facter. For example to access the 'ipaddress' fact from the output, specify @host.facts['ipaddress'].
@host.grub_pass	Returns the host's GRUB password.
@host.hostgroup	The host group of the host.

Name	Description
host_enc['parameters']	Returns a Ruby hash containing information on host parameters. For example, use host_enc['parameters'] ['lifecycle_environment'] to get the life cycle environment of a host.
@host.image_build?	Returns true if the host is provisioned using an image.
@host.interfaces	Contains an array of all available host interfaces including the primary interface. See Note .
@host.interfaces_with_identifier('IDs')	Returns array of interfaces with given identifier. You can pass an array of multiple identifiers as an input, for example @host.interfaces_with_identifier(['eth0', 'eth1']). See Note .
@host.ip	The IP address of the host.
@host.location	The location of the host.
@host.mac	The MAC address of the host.
@host.managed_interfaces	Returns an array of managed interfaces (excluding BMC and bonded interfaces). See Note .
@host.medium	The assigned operating system installation medium.
@host.name	The full name of the host.
@host.operatingsystem.family	The operating system family.
@host.operatingsystem.major	The major version number of the assigned operating system.
@host.operatingsystem.minor	The minor version number of the assigned operating system.
@host.operatingsystem.name	The assigned operating system name.
@host.operatingsystem.boot_files_uri(@host.medium,@host.architecture)	Full path to the kernel and initrd, returns an array.
@host.os.medium_uri(@host)	The URI used for provisioning (path configured in installation media).
host_param('parameter_name')	Returns the value of the specified host parameter.

Name	Description
host_param_false?('parameter_name')	Returns false if the specified host parameter evaluates to false.
host_param_true?('parameter_name')	Returns true if the specified host parameter evaluates to true.
@host.primary_interface	Returns the primary interface of the host.
@host.provider	The compute resource provider.
@host.provision_interface	Returns the provisioning interface of the host. Returns an interface object.
@host.ptable	The partition table name.
@host.puppetmaster	The Puppet master the host should use.
@host.pxe_build?	Returns true if the host is provisioned using the network or PXE.
@host.shortname	The short name of the host.
@host.sp_ip	The IP address of the BMC interface.
@host.sp_mac	The MAC address of the BMC interface.
@host.sp_name	The name of the BMC interface.
@host.sp_subnet	The subnet of the BMC network.
@host.subnet.dhcp	Returns true if a DHCP proxy is configured for this host.
@host.subnet.dns_primary	The primary DNS server of the host.
@host.subnet.dns_secondary	The secondary DNS server of the host.
@host.subnet.gateway	The gateway of the host.
@host.subnet.mask	The subnet mask of the host.
@host.url_for_boot(:initrd)	Full path to the initrd image associated with this host. Not recommended, as it does not interpolate variables.

Name	Description
@host.url_for_boot(:kernel)	Full path to the kernel associated with this host. Not recommended, as it does not interpolate variables, prefer <code>boot_files_uri</code> .
@provisioning_type	Equals to 'host' or 'hostgroup' depending on type of provisioning.
@static	Returns true if the network configuration is static.
@template_name	Name of the template being rendered.
grub_pass	Returns the GRUB password wrapped in md5pass argument, that is --md5pass=#{@host.grub_pass} .
ks_console	Returns a string assembled using the port and the baud rate of the host which can be added to a kernel line. For example console=ttys1,9600 .
root_pass	Returns the root password configured for the system.

NOTE

Host variables related to network interfaces, such as `@host.interfaces` or `@host.bond_interfaces` return interface data grouped in an array. To extract a parameter value of a specific interface, use Ruby methods to parse the array. For example, to get information about the first interface from an array and use it in a kickstart template:

```
<% myinterface = @host.interfaces.first %>
IPADDR="<%= myinterface.ip %>"
NETMASK="<%= myinterface.subnet.mask %>"
GATEWAY="<%= myinterface.subnet.gateway %>"
```

You can iterate over the interface array, for example to extract an array of interface names use:

```
<% ifnames = []
@host.interfaces.each do |i|
  ifnames.push(i.name)
end %>
```

Example A.2. Using Host Specific Variables

The following example checks if the host has Puppet and the Puppetlabs repository enabled:

```
<%
pm_set = @host.puppetmaster.empty? ? false : true
puppet_enabled = pm_set || host_param_true?('force-puppet')
```

```
puppetlabs_enabled = host_param_true?('enable-puppetlabs-repo')
%>
```

The following example shows how to capture the minor and major version of the host's operating system, which can be used for package related decisions:

```
<%
os_major = @host.operatingsystem.major.to_i
os_minor = @host.operatingsystem.minor.to_i
%>

<% if ((os_minor < 2) && (os_major < 14)) -%>
...
<% end -%>
```

The following example imports the 'kickstart_networking_setup' snippet if the host's subnet has the DHCP boot mode enabled:

```
<% subnet = @host.subnet %>
<% if subnet.respond_to?(:dhcp_boot_mode?) -%>
<%= snippet 'kickstart_networking_setup' %>
<% end -%>
```

The majority of common Ruby methods can be applied on host specific variables. For example, to extract the last segment of the host's IP address, you can use:

```
<% @host.ip.split('.').last %>
```

The following variables are designed to be used within kickstart provisioning templates.

Table A.4. Kickstart Specific Variables

Name	Description
@arch	The host architecture name, same as @host.architecture.name.
@dynamic	Returns true if the partition table being used is a %pre script (has the #Dynamic option as the first line of the table).
@epel	A command which will automatically install the correct version of the epel-release rpm. Use in a %post script.
@mediapath	The full kickstart line to provide the URL command.
@osver	The operating system major version number, same as @host.operatingsystem.major.

APPENDIX B. HOST MANAGEMENT WITHOUT GOFERD

From **Satellite 6.2.11** onward, errata and package management via remote execution is available using a **yum** plugin. This allows the goferd service daemon to be disabled, thereby reducing memory and CPU load on content hosts.

The **yum** plugin is included with the **katello-host-tools**. This shipped with the Satellite 6.2.11 client update.

B.1. PREREQUISITES

The following must be made on all content hosts to allow host management by remote execution.

- Ensure **katello-agent** is installed on the content host by following [Section 2.9.4, “Installing the Katello Agent”](#).
- Stop the goferd service:

```
# systemctl stop goferd.service
```
- Disable the goferd service:

```
# systemctl disable goferd.service
```
- Distribute the SSH keys to the content hosts by following [Section 5.1, “Establishing a Secure Connection for Remote Commands”](#).

B.2. CONFIGURING HOST MANAGEMENT WITHOUT GOFERD AS THE SYSTEM DEFAULT

These steps configure host management to use remote execution as the system default for future package deployments.

To Configure Host Management Without Goferd as the System Default:

1. Log in to the Satellite web UI.
2. Navigate to **Administer > Settings**.
3. Select the **Content** tab.
4. Set the **Use remote execution by default** parameter to **Yes**.

The Satellite server now uses host management by remote execution instead of goferd.

B.3. LIMITATIONS WITH HAMMER

The following applies if you are using the **hammer** command to push errata. The **hammer** command is dependent on goferd to manage errata on content hosts. As a workaround, use Satellite’s remote execution feature to apply errata.

To Use Hammer Remote Execution Commands:

For example, perform a **yum -y update** on *host123.example.org*:

```
# hammer job-invocation create \  
--job-template "Run Command - SSH Default" \  
--inputs command="yum -y update" \  
--search-query "name ~ host123"  
Job invocation 24 created  
[.....] [100%]  
1 task(s), 1 success, 0 fail
```