Red Hat Enterprise Linux 8

Configuring basic system settings

A guide to configuring basic system settings on Red Hat Enterprise Linux 8
Red Hat Enterprise Linux 8 Configuring basic system settings

A guide to configuring basic system settings on Red Hat Enterprise Linux 8
Abstract

This document describes basics of system administration on Red Hat Enterprise Linux 8. The title focuses on: basic tasks that a system administrator needs to do just after the operating system has been successfully installed, installing software with yum, using systemd for service management, managing users, groups and file permissions, using chrony to configure NTP, working with Python 3 and others.
# Table of Contents

**CHAPTER 1. GETTING STARTED WITH SYSTEM ADMINISTRATION IN RED HAT ENTERPRISE LINUX**  
1.1. WHAT THE RHEL 8 WEB CONSOLE IS AND WHICH TASKS IT CAN BE USED FOR  
1.2. CONFIGURING SYSTEM SETTINGS IN THE WEB CONSOLE  
  1.2.1. Using the web console to restart the system  
    Prerequisites  
    Procedure  
  1.2.2. Using the web console to shutdown the system  
    Prerequisites  
    Procedure  
  1.2.3. Using the web console for setting a host name  
    Prerequisites  
    Procedure  
  1.2.4. Joining the RHEL 8 system to the IdM domain using the web console  
    Prerequisites  
    Procedure  
  1.2.5. Using the web console for configuring time settings  
    Prerequisites  
    Procedure  
  1.2.6. Using the web console for selecting performance profiles  
    Prerequisites  
    Procedure  
1.3. WHAT RHEL SYSTEM ROLES ARE AND WHICH TASKS THEY CAN BE USED FOR  
  1.3.1. Introduction to RHEL System Roles  
  1.3.2. Applying a role  
    Prerequisites  
    Procedure  
1.4. BASIC CONFIGURATION OF ENVIRONMENT  
  1.4.1. Introduction to configuring the date and time  
  1.4.2. Introduction to configuring the system locale  
  1.4.3. Introduction to configuring the keyboard layout  
1.5. CONFIGURING AND MANAGING NETWORK ACCESS  
  1.5.1. Configuring network access during the installation process  
  1.5.2. Managing network connections after the installation process using nmcli  
  1.5.3. Managing network connections after the installation process using nmtui  
  1.5.4. Managing networking in the RHEL 8 web console  
  1.5.5. Managing networking using RHEL System Roles  
1.6. REGISTERING THE SYSTEM AND MANAGING SUBSCRIPTIONS  
  1.6.1. Registering the system after the installation  
  1.6.2. Registering subscriptions with credentials in the web console  
    Prerequisites  
    Procedure  
1.7. INSTALLING SOFTWARE  
  1.7.1. Prerequisites for software installation  
  1.7.2. Introduction to the system of software packaging and software repositories  
  1.7.3. Managing basic software-installation tasks with subscription manager and yum  
1.8. MAKING SYSTEMD SERVICES START AT BOOT TIME  
  1.8.1. Enabling or disabling the services  
  1.8.2. Managing services in the RHEL 8 web console  
1.9. ENHANCING SYSTEM SECURITY WITH A FIREWALL, SELINUX AND SSH LOGINGS  

---
1.9.1. Ensuring the firewall is enabled and running
1.9.1.1. What a firewall is and how it enhances system security
1.9.1.2. Re-enabling the firewalld service
1.9.1.3. Managing firewall in the RHEL 8 web console
1.9.2. Ensuring the appropriate state of SELinux
1.9.2.1. What SELinux is and how it enhances system security
1.9.2.2. SELinux states
1.9.2.3. SELinux modes
1.9.3. Ensuring the Required State of SELinux
1.9.3.1. Managing SELinux in the RHEL 8 web console
1.9.4. Managing SELinux using RHEL System Roles
1.9.5. Accessing system through SSH
1.9.5.1. What SSH-based access is and how it enhances system security
1.9.5.2. Configuring key-based SSH access
1.9.5.3. Disabling SSH root login
1.10. MANAGING USER ACCOUNTS
1.10.1. The basics of managing user accounts
1.10.1.1. Normal and System Accounts
1.10.1.2. What groups are and which purposes they can be used for
1.10.1.3. Basic command-line tools to manage user accounts and groups
1.10.2. System user accounts managed in the web console
1.10.3. Adding new accounts in the web console
Procedure
1.11. DUMPING THE CRASHED KERNEL USING THE KDUMP MECHANISM
1.11.1. What kdump is and which tasks it can be used for
1.11.2. Ensuring that kdump is installed and enabled after the installation process
1.11.3. Configuring kdump in the RHEL 8 web console
1.11.4. Configuring kdump using RHEL System Roles
1.12. PERFORMING SYSTEM RESCUE AND CREATING SYSTEM BACKUP WITH REAR
1.12.1. What ReaR is and which tasks it can be used for
1.12.2. Quickstart to installation and configuration of ReaR
1.12.3. Quickstart to creation of the rescue system with ReaR
1.12.4. Quickstart to configuration of ReaR with the backup software
1.13. USING THE LOG FILES TO TROUBLESHOOT PROBLEMS
1.13.1. Services handling the syslog messages
1.13.2. Subdirectories storing the syslog messages
1.13.2.1. Managing the log files in the RHEL 8 web console
1.14. ACCESSING RED HAT SUPPORT
1.14.2. Using the SOS report to troubleshoot problems

CHAPTER 2. INSTALLING SOFTWARE WITH YUM

2.1. INTRODUCTION TO INSTALLING SOFTWARE ON RED HAT ENTERPRISE LINUX 8
2.1.1. Application streams
2.2. INTRODUCTION TO YUM FUNCTIONALITY
2.3. USING YUM FOR PARTICULAR TASKS
2.3.1. Checking for updates and updating packages
2.3.1.1. Checking for updates
2.3.1.1.1. Updating packages
2.3.1.1.1.1. Updating a single package
2.3.1.1.1.2. Updating a package group
2.3.1.1.2. Updating all packages and their dependencies
2.3.1.1.3. Updating security-related packages
2.3.2. Working with packages
  2.3.2.1. Searching packages
  2.3.2.1.1. Filtering the Results
  2.3.2.2. Listing packages
  2.3.2.2.1. Listing repositories
  2.3.2.2.2. Displaying package information
  2.3.2.2.3. Installing packages
  2.3.2.2.4. Removing packages
2.3.3. Working with package groups
  2.3.3.1. Listing package groups
  2.3.3.2. Installing a package group
  2.3.3.3. Removing a package group
2.4. WORKING WITH TRANSACTION HISTORY
  2.4.1. Listing transactions
  2.4.2. Reverting and repeating transactions
2.5. CONFIGURING YUM AND YUM REPOSITORIES
  2.5.1. Viewing the current configuration
  2.5.2. Setting [main] options
  2.5.3. Setting [repository] options
  2.5.4. Adding, enabling, and disabling a yum repository
    2.5.4.1. Adding a yum repository
    2.5.4.2. Enabling a yum repository
    Disabling a yum repository
2.6. USING YUM PLUG-INS
  2.6.1. Enabling, configuring, and disabling yum plug-ins
2.7. ADDITIONAL RESOURCES
  2.7.1. Installed Documentation
  2.7.2. Online Documentation

CHAPTER 3. MANAGING SERVICES WITH SYSTEMD

3.1. INTRODUCTION TO SYSTEMD
  Overriding the default systemd configuration using system.conf
  3.1.1. Main features
  3.1.2. Compatibility changes
3.2. MANAGING SYSTEM SERVICES
  Specifying service units
  Behavior of systemctl in a chroot environment
  3.2.1. Listing services
  3.2.2. Displaying service status
  3.2.3. Starting a service
  3.2.4. Stopping a service
  3.2.5. Restarting a service
  3.2.6. Enabling a service
  3.2.7. Disabling a service
  3.2.8. Starting a conflicting service
3.3. WORKING WITH SYSTEMD TARGETS
  3.3.1. Viewing the default target
  3.3.2. Viewing the current target
  3.3.3. Changing the default target
  3.3.4. Changing the current target
  3.3.5. Changing to rescue mode
  3.3.6. Changing to emergency mode
3.4. SHUTTING DOWN, SUSPENDING, AND HIBERNATING THE SYSTEM
3.4.1. Shutting down the system
Using systemctl commands
Using the shutdown command
3.4.2. Restarting the system
3.4.3. Suspending the system
3.4.4. Hibernating the system

3.5. WORKING WITH SYSTEMD UNIT FILES
3.5.1. Understanding the unit file structure
3.5.2. Creating custom unit files
3.5.3. Converting SysV init scripts to unit files
Finding the service description
Finding service dependencies
Finding default targets of the service
Finding files used by the service
3.5.4. Modifying existing unit files
Extending the default unit configuration
Overriding the default unit configuration
Monitoring overridden units
3.5.5. Working with instantiated units

3.6. OPTIMIZING SYSTEMD TO SHORTEN THE BOOT TIME
3.6.1. Examining system boot performance
Prerequisites
Analyzing overall boot time
Procedure
Analyzing unit initialization time
Procedure
Identifying critical units
Procedure
3.6.2. A guide to selecting services that can be safely disabled

3.7. ADDITIONAL RESOURCES
3.7.1. Installed Documentation
3.7.2. Online Documentation

CHAPTER 4. MANAGING USER AND GROUP ACCOUNTS AND SETTING PERMISSIONS ON FILES
4.1. INTRODUCTION TO USERS AND GROUPS
4.2. RESERVED USER AND GROUP IDS
4.3. USER PRIVATE GROUPS
4.4. SHADOW PASSWORDS
4.5. MANAGING USERS IN A GRAPHICAL ENVIRONMENT
4.5.1. Using the Users Settings tool
4.6. MANAGING USERS USING COMMAND-LINE TOOLS
4.6.1. Adding a new user
4.6.2. Adding a new group
4.6.3. Adding an existing user to an existing group
4.6.4. Creating group directories
4.6.5. Setting default permissions for new files using umask
4.6.5.1. What umask consists of
4.6.5.2. How umask works
4.6.5.3. Managing umask in Shells
4.6.5.3.1. Displaying the current mask
4.6.5.3.2. Setting mask in shell using umask
4.6.5.3.3. Working with the default shell umask
4.6.5.3.4. Working with the default shell umask of a specific user
6.2.1.3. Naming conventions for Python 3 packages 135
6.2.2. Installing and using Python 2 136
  6.2.2.1. Installing Python 2 136
  6.2.2.2. Using Python 2 136
    6.2.2.2.1. Naming conventions for Python 2 packages 136
6.2.3. Configuring the unversioned Python 136
6.3. MIGRATING FROM PYTHON 2 TO PYTHON 3 137
6.4. PACKAGING OF PYTHON 3 RPMS 137
  6.4.1. Typical SPEC file description for a Python RPM package 137
  6.4.2. Common macros for Python 3 RPM packages 139
  6.4.3. Automatic provides for Python RPM packages 140
  6.4.4. Handling hashbangs in Python scripts 140

CHAPTER 7. INSTALLING AND USING LANGPACKS ................................. 142

  7.1. INTRODUCTION TO LANGPACKS 142
  7.2. WORKING WITH RPM WEAK DEPENDENCY-BASED LANGPACKS 142
    7.2.1. Querying RPM weak dependency-based langpacks 142
    7.2.2. Installing language support 142
    7.2.3. Removing language support 143
  7.3. SAVING DISK SPACE BY USING GLIBC-LANGPACK-<LOCALE_CODE> 143

CHAPTER 8. GETTING STARTED WITH TCL/TK ON RED HAT ENTERPRISE LINUX 8 .................. 144

  8.1. INTRODUCTION TO TCL/TK 144
  8.2. NOTABLE CHANGES IN TCL/TK 8.6 144
  8.3. MIGRATING TO TCL/TK 8.6 145
    8.3.1. Migration path for developers of Tcl extensions 145
    8.3.2. Migration path for users scripting their tasks with Tcl/Tk 145

CHAPTER 9. USING PREFIXDEVNAME FOR NAMING OF ETHERNET NETWORK INTERFACES .......... 147

  9.1. INTRODUCTION TO PREFIXDEVNAME 147
  9.2. SETTING PREFIXDEVNAME 147
  9.3. LIMITATIONS OF PREFIXDEVNAME 147
CHAPTER 1. GETTING STARTED WITH SYSTEM ADMINISTRATION IN RED HAT ENTERPRISE LINUX

The following sections provide an overview of the basic tasks that system administrators might need to perform just after Red Hat Enterprise Linux has been installed.

NOTE

Such tasks may include items that are usually done already during the installation process, but they do not have to be done necessarily, such as the registration of the system. The sections dealing with such tasks provide a brief summary of how this can be achieved during the installation and links to related documentation.

For information on Red Hat Enterprise Linux installation, see Performing a standard RHEL installation.

NOTE

The following sections mention some commands to be performed. The commands that need to be entered by the root user have # in the prompt, while the commands that can be performed by a regular user, have $ in their prompt.

Although all post-installation tasks can be achieved through the command line, you can also use the RHEL 8 web console to perform some of them.

1.1. WHAT THE RHEL 8 WEB CONSOLE IS AND WHICH TASKS IT CAN BE USED FOR

The RHEL 8 web console is an interactive server administration interface. It interacts directly with the operating system from a real Linux session in a browser.

The web console enables to perform these tasks:

- Monitoring basic system features, such as hardware information, time configuration, performance profiles, connection to the realm domain
- Inspecting system log files
- Managing network interfaces and configuring firewall
- Handling docker images
- Managing virtual machines
- Managing user accounts
- Monitoring and configuring system services
- Creating diagnostic reports
- Setting kernel dump configuration
- Managing packages
1.2. CONFIGURING SYSTEM SETTINGS IN THE WEB CONSOLE

In this chapter, you will learn how to execute basic system settings in the web console and thus be able to:

- Restart or shutdown the system in the web console.
- Change a system host name.
- Join the system to a domain.
- Configure time and time zones.
- Change a performance profile.

1.2.1. Using the web console to restart the system

The following procedure describes system restart executed in the web console.

Prerequisites

- The web console must be installed and accessible.
  For details, see Installing the web console.

Procedure

1. Log in to the RHEL 8 web console.
   For details, see Logging in to the web console.

2. Click System.

3. In the Power Options drop down list, select Restart.
4. If there are users logged into the system, write a reason for the restart in the **Restart** dialog box.

5. In the **Delay** drop down list, select a time interval.

6. Click **Restart**.

   The system will be restarted according to your selection.
1.2.2. Using the web console to shutdown the system

The following procedure describes system shutdown executed in the web console.

Prerequisites

- The web console must be installed and accessible.
  For details, see Installing the web console.

Procedure

1. Log in to the RHEL 8 web console.
   For details, see Logging in to the web console.
2. Click System.
3. In the Power Options drop down list, select Shut Down.
4. If there are users logged into the system, write a reason for the shutdown in the Shut Down dialog box.
5. In the Delay drop down list, select a time interval.
6. Click Shut Down.

The system will be turned off according to your selection.

1.2.3. Using the web console for setting a host name

The host name identifies the system. By default, the host name is set to localhost, but you can change it.
Host names consist of two parts:

- Host name – It is a unique name which identifies a system.
- Domain – If you want to use the machine in the network and use names instead of just IP addresses, you need to add the domain as a suffix behind the host name. For example: `mymachine.example.com`

You can configure also a pretty host name in the RHEL web console. The pretty host name allows you to enter a host name with capital letters, spaces, and so on. The pretty host name displays in the web console, but it does not have to correspond with the host name.

Example:

Pretty host name: My Machine Host name: `mymachine` Real host name (Fully qualified domain name): `mymachine.idm.company.com`

Host names are stored in the `/etc/hostname` file, however, you can set or change the host name in the web console.

**Prerequisites**

- The web console must be installed and accessible.
  For details, see **Installing the web console**.

**Procedure**

1. Log in to the RHEL 8 web console.
   For details, see **Logging in to the web console**.

2. Click **System**.

3. Click the current host name.

4. In the **Change Host Name** dialog box, enter the host name in the **Pretty Host Name** field.

5. In the **Real Host Name** field, the pretty name will be compounded with a domain name.
   You can change the host name manually if it does not correspond with the pretty host name.

6. Click **Change**.
To verify that the host name is configured properly, try to log out from the web console and add to the browser the address with the new host name.

1.2.4. Joining the RHEL 8 system to the IdM domain using the web console

The following procedure describes joining the RHEL 8 system to the IdM domain.

**Prerequisites**

- IdM domain running and reachable from the client you want to join.
- IdM domain administrator credentials.

**Procedure**

1. Log in to the RHEL web console. For details, see Logging in to the web console.
2. Open the **System** tab.
3. Click **Join Domain**.
4. In the **Join a Domain** dialog box, enter the host name of the IdM server in the **Domain Address** field.

5. In the **Authentication** drop down list, select if you want to use password or one time password for authentication.

6. In the **Domain Administrator Name** field, enter the user name of the IdM administration account.

7. In the password field, add the password or one time password according to what you selected in the **Authentication** drop down list above.

8. Click **Join**.
If the RHEL 8 web console did not display an error, the system has been joined to the IdM domain and you can see the domain name in the **System** screen.

![System Screen](image)

**WARNING**

If you click to the joined domain in the **System** screen, the system will display a warning dialog with the information about leaving the domain. If you click **Leave**, the system will leave the domain.
1.2.5. Using the web console for configuring time settings

This section shows you how to set:

- The correct time zone
- Automatic time settings provided by an NTP server.
- A specific NTP server.

Prerequisites

- The web console must be installed and accessible. For details, see Installing the web console.

Procedure

1. Log in to the RHEL 8 web console. For details, see Logging in to the web console.
2. Click System.
3. Click the current system time.
4. In the Change System Time dialog box, change the time zone if necessary.
5. In the Set Time drop down menu, select:
   - Manually
• Automatically using NTP server – This is a default option. If the time of the system is correct, leave it as it is.

• Automatically using specific NTP servers – Use this option only if you need to synchronize the system with a specific NTP server and add the DNS name or IP address of the server.

6. Click **Change**.

   ![Change System Time](image)

   The change is now available in the **System** tab.

### 1.2.6. Using the web console for selecting performance profiles

Red Hat Enterprise Linux 8 includes performance profiles optimizing:

- Systems using Desktop
- Latency performance
- Network performance
- Low power consumption
- Virtual machines

The following procedure describes setting up performance profiles in the web console.

The RHEL 8 web console configures the **tuned** service.

For details about the **tuned** service, see [Monitoring and managing system status and performance](#).

**Prerequisites**

- The web console must be installed and accessible.
  - For details, see [Installing the web console](#).

**Procedure**

1. Log in to the RHEL 8 web console.
  - For details, see [Logging in to the web console](#).
2. Click **System**.

3. In the **Performance Profile** field, click the current performance profile.

4. In the **Change Performance Profile** dialog box, change the profile if necessary.

5. Click **Change**.

The change is now available in the **System** tab.

### 1.3. WHAT RHEL SYSTEM ROLES ARE AND WHICH TASKS THEY CAN BE USED FOR

#### 1.3.1. Introduction to RHEL System Roles

Red Hat Enterprise Linux System Roles is a collection of Ansible roles and modules that provide a configuration interface to remotely manage multiple RHEL systems. The interface enables managing system configurations across multiple versions of RHEL, as well as adopting new major releases.

Red Hat Enterprise Linux System Roles were introduced with Red Hat Enterprise Linux 7.4. For more information, see the [Red Hat Enterprise Linux (RHEL) System Roles](https://access.redhat.com/documentation/en-us/red_hat_enterprise_linux/7/html/system_roles) Red Hat KnowledgeBase article.

On Red Hat Enterprise Linux 8, the interface currently consists of the following roles:
All these roles are provided by the `rhel-system-roles` package available in the AppStream repository.

### 1.3.2. Applying a role

To apply a particular role, you need to fulfill the following prerequisites.

#### Prerequisites

- The `rhel-system-roles` package has been installed on the system that you want to use as a control node:

  ```
  ~]$ yum install rhel-system-roles
  ```

- The Ansible Engine repository has been enabled, and the `ansible` package has been installed on the system that you want to use as a control node. The `ansible` package is needed to run playbooks that use Red Hat Enterprise Linux System Roles.

  **If you do not have a Red Hat Ansible Engine Subscription, you can use a limited supported version of Red Hat Ansible Engine provided with your Red Hat Enterprise Linux subscription.** In this case, follow these steps:

  1. Enable the RHEL Ansible Engine repository:

     ```
     ~]$ subscription-manager refresh
     ~]$ subscription-manager repos --enable ansible-2-for-rhel-8-x86_64-rpms
     ```

  2. Install Ansible Engine:

     ```
     ~]$ yum install ansible
     ```

    **If you have a Red Hat Ansible Engine Subscription, follow the procedure described in [How do I Download and Install Red Hat Ansible Engine?](#).**

- You are able to create an Ansible playbook. Playbooks represent Ansible’s configuration, deployment, and orchestration language. By using playbooks, you can declare and manage configurations of remote machines, deploy multiple remote machines or orchestrate steps of any manual ordered process.

  A playbook is a list of one or more **plays**. Every **play** can include Ansible variables, tasks or roles.

  Playbooks are human-readable, and they are expressed in the **YAML** format.

  For more information about playbooks, see [Ansible documentation](#).
The classic way to use roles is via the `roles:` option for a given `play`, as shown in the example below:

```yaml
---
- hosts: webservers
  roles:
    - rhel-system-roles.network
    - rhel-system-roles.timesync
```

For more information on using roles in playbooks, see [Ansible documentation](https://docs.ansible.com/ansible/latest/). See [Ansible examples](https://docs.ansible.com/ansible/latest/examples.html) for example playbooks.

**NOTE**

Every role includes a README file, which documents how to use the role and supported parameter values. You can also find an example playbook for a particular role under the documentation directory of the role. Such documentation directory is provided by default with the `rhel-system-roles` package, and can be found in the following location:

```
/usr/share/doc/rhel-system-roles-<version>/SUBSYSTEM/
```

where SUBSYSTEM is the name of the subsystem that contains the individual role manages - selinux, kdump, network or timesync.

2. Execute the playbook on targeted hosts by running the `ansible-playbook` command:

```
ansible-playbook -i <name of the inventory> <name of the playbook>
```

An inventory is a list of systems against which Ansible works. For more information on how to create and inventory, and how to work with it, see [Ansible documentation](https://docs.ansible.com/ansible/latest/).

If you have not created an inventory in advance, you can do so even at the time of running `ansible-playbook`:

For cases with only one targeted host against which you want to run the playbook, use:

```
ansible-playbook -i host1, <name of the playbook>
```

For cases with multiple targeted hosts against which you want to run the playbook, use:

```
ansible-playbook -i host1,host2,....,hostn <name of the playbook>
```

For more detailed information on using the `ansible-playbook` command, see the `ansible-playbook` man page.

**1.4. BASIC CONFIGURATION OF ENVIRONMENT**

Basic configuration of environment includes:

- Date and time
1.4.1. Introduction to configuring the date and time

Accurate timekeeping is important for a number of reasons. In Red Hat Enterprise Linux, timekeeping is ensured by the NTP protocol, which is implemented by a daemon running in user space. The user space daemon updates the system clock running in the kernel. The system clock can keep time by using various clock sources.

Red Hat Enterprise Linux 8 uses the chronyd daemon to implement NTP. chronyd is available from the chrony package. For more information on configuring and using NTP with chronyd, see Chapter 5, Using the Chrony suite to configure NTP.

Displaying the current date and time

- To display the current date and time, use one of the following commands:
  
  ```
  ~]$ date
  ~]$ timedatectl
  ```

  Note that the `timedatectl` command provides more verbose output, including universal time, currently used time zone, the status of the Network Time Protocol (NTP) configuration, and some additional information.

For more information on configuring the date and time during the installation, see Performing a standard RHEL installation.

1.4.1.1. Managing time configurations in the RHEL 8 web console

Time settings in the web console describes Using the web console for configuring time settings.

1.4.2. Introduction to configuring the system locale

System-wide locale settings are stored in the `/etc/locale.conf` file, which is read at early boot by the systemd daemon. The locale settings configured in `/etc/locale.conf` are inherited by every service or user, unless individual programs or individual users override them.

Basic tasks to handle the system locales:

- Listing available system locale settings:
  
  ```
  ~]$ localectl list-locales
  ```

- Displaying current status of the system locales settings:
  
  ```
  ~]$ localectl status
  ```

- Setting or changing the default system locale settings:
  
  ```
  ~]$ localectl set-locale LANG=locale
  ```
1.4.3. Introduction to configuring the keyboard layout

The keyboard layout settings control the layout used on the text console and graphical user interfaces.

Basic tasks to handle the keyboard layout include:

- Listing available keymaps:
  ```
  ~]$ localectl list-keymaps
  ```

- Displaying current status of keymap settings:
  ```
  ~]$ localectl status
  ```

- Setting or changing the default system keymap:
  ```
  ~]$ localectl set-keymap
  ```

1.5. CONFIGURING AND MANAGING NETWORK ACCESS

1.5.1. Configuring network access during the installation process

Ways to configure network access during the installation process:

- The **Network & Hostname** menu at the Installation Summary screen in the graphical user interface of the **Anaconda** installation program

- The **Network settings** option in the text mode of the **Anaconda** installation program

- The Kickstart file

When the system boots for the first time after the installation has finished, any network interfaces which you configured during the installation are automatically activated.

For detailed information on configuration of network access during installation process, see [Installing Red Hat Enterprise Linux 8](#).

1.5.2. Managing network connections after the installation process using **nmcli**

Run the following commands to manage network connections using the **nmcli** utility.

**NOTE**

The **nmcli** utility has a powerful syntax completion feature when the **Tab** key is pressed twice. You need to have the **bash-completion** package installed to enable it.

To create a new connection:

```
~]$ nmcli con add type of the connection con-name connection name ifname ifname ipv4.addresses ipv4 address ipv4.gateway gateway address
```
Here, replace:

- **type of the connection** by the required type of the device
- **connection name** by the required connection name
- **ifname** by the required device name
- **ipv4 address** by the required IPv4 address/netmask
- **gateway address** by the required gateway address

Note that **ipv4 address** and **gateway address** are optional settings, while all remaining settings are required.

You can also create a new connection in assisted mode. To do so, run this command, and follow the instructions that will prompt you for input of particular configuration settings of this connection:

```
~# nmcli -a con add
```

To modify the existing connection:

```
~# nmcli con mod connection_name setting.property newvalue
```

Here, replace:

- **connection name** by the name of the connection that you want to modify
- **setting.property** by the configuration setting that you want to modify
- **newvalue** by the required value of this configuration setting

For example, to set the method of the configuration of IPv4 address (**ipv4.method**) to **auto** for the connection named **enp0**, use the following command:

```
~# nmcli con mod enp0 ipv4.method auto
```

To edit a connection, run the following command:

```
~# nmcli connection edit connection_name
```

Here, replace **connection name** by the name of the connection that you want to edit.

To display all connections:

```
~# nmcli con show
```

To display active connections:

```
~# nmcli con show --active
```

To display all configuration settings of a particular connection:

```
~# nmcli con show con-name connection_name
```
Here, replace \textit{connection name} by the name of the required connection.

Then, follow the instructions that will prompt you for input of particular configuration settings. To display all possible options of any configuration setting, use the \texttt{print} command in the editor.

1.5.3. Managing network connections after the installation process using \texttt{nmtui}

The \texttt{NetworkManager} text user interface (TUI) utility, \texttt{nmtui}, provides a text interface to configure networking by controlling \texttt{NetworkManager}.

1.5.4. Managing networking in the RHEL 8 web console

in the web console, the \texttt{Networking} menu enables you:

- To display currently received and sent packets
- To display the most important characteristics of available network interfaces
- To display content of the networking logs.
- To add various types of network interfaces (bond, team, bridge, VLAN)

![Figure 1.1. Managing Networking in the RHEL 8 web console](image)

1.5.5. Managing networking using RHEL System Roles

You can configure the networking connections on multiple target machines using the \texttt{network} role.

The \texttt{network} role allows to configure the following types of interfaces:

- Ethernet
- Bridge
- Bonded
The required networking connections for each host are provided as a list within the `network_connections` variable.

### WARNING

The `network` role updates or creates all connection profiles on the target system exactly as specified in the `network_connections` variable. Therefore, the `network` role removes options from the specified profiles if the options are only present on the system but not in the `network_connections` variable.

The following example shows how to apply the `network` role to ensure that an Ethernet connection with the required parameters exists:

```yaml
Example 1.1. An example playbook applying the network role to set up an Ethernet connection with the required parameters

```# SPDX-License-Identifier: BSD-3-Clause
---
- hosts: network-test
  vars:
    network_connections:
      # Create one ethernet profile and activate it.
      # The profile uses automatic IP addressing and is tied to the interface by MAC address.
      - name: prod1
        state: up
        type: ethernet
        autoconnect: yes
        mac: "00:00:5e:00:53:00"
        mtu: 1450

  roles:
    - rhel-system-roles.network
```

For more information on applying the `network` role, and for more examples on using the role, see the System roles documentation.

### 1.6. REGISTERING THE SYSTEM AND MANAGING SUBSCRIPTIONS

The products installed on Red Hat Enterprise Linux, including the operating system itself, are covered by subscriptions.
A subscription to Red Hat Content Delivery Network is used to track:

- Registered systems
- Products installed on those systems
- Subscriptions attached to those products

### 1.6.1. Registering the system after the installation

Your subscription can be registered during the installation process. For more information, see Performing a standard RHEL installation.

If you have not registered your system during the installation process, you can do it afterwards by applying the following procedure. Note that all commands in this procedure need to be performed as the root user.

**Registering and subscribing your system**

1. Register your system:

   ```
   ~]# subscription-manager register
   ```

   The command will prompt you to enter your Red Hat Customer Portal user name and password.

2. Determine the pool ID of a subscription that you require:

   ```
   ~]# subscription-manager list --available
   ```

   This command displays all available subscriptions for your Red Hat account. For every subscription, various characteristics are displayed, including the pool ID.

3. Attach the appropriate subscription to your system by replacing `pool_id` with the pool ID determined in the previous step:

   ```
   ~]# subscription-manager attach --pool=pool_id
   ```

### 1.6.2. Registering subscriptions with credentials in the web console

The following describes subscribing the newly installed Red Hat Enterprise Linux using the RHEL 8 web console.

**Prerequisites**

- Valid user account in the Red Hat Customer Portal. See the Create a Red Hat Login page.
- Active subscription for the RHEL system.

**Procedure**

1. Type subscription in the search field and press the Enter key.
Alternatively, you can log in to the RHEL 8 web console. For details, see Logging in to the web console.

2. In the polkit authentication dialog for privileged tasks, add the password belonging user name displayed in the dialog.

3. Click **Authenticate**.

4. In the **Subscriptions** dialog box, click **Register**.

   ![Subscriptions dialog box](image)

5. Enter your Customer Portal credentials.
6. Enter the name of your organization.
   You need to add the organization name or organization ID, if you have more than one account in the Red Hat Customer Portal. To get the org ID, go to your Red Hat contact point.

7. Click the **Register** button.

At this point, your RHEL 8 system has been successfully registered.
1.7. INSTALLING SOFTWARE

This section provides information to guide you through the basics of software installation. It mentions the prerequisites that you need to fulfil to be able to install software, provides the basic information on software packaging and software repositories, and references the ways to perform basic tasks related to software installation.

1.7.1. Prerequisites for software installation

The Red Hat Content Delivery Network subscription service provides a mechanism to handle Red Hat software inventory and enables you to install additional software or update already installed packages. You can start installing software once you have registered your system and attached a subscription, as described in Section 1.6.1, “Registering the system after the installation”.

1.7.2. Introduction to the system of software packaging and software repositories

All software on a Red Hat Enterprise Linux system is divided into RPM packages, which are stored in particular repositories. When a system is subscribed to the Red Hat Content Delivery Network, a repository file is created in the /etc/yum.repos.d/ directory.
Use the **yum** utility to manage package operations:

- Searching information about packages
- Installing packages
- Updating packages
- Removing packages
- Checking the list of currently available repositories
- Adding or removing a repository
- Enabling or disabling a repository

For information on basic tasks related to the installation of software, see Section 1.7.3, "Managing basic software-installation tasks with subscription manager and yum".

### 1.7.3. Managing basic software-installation tasks with subscription manager and yum

The most basic software-installation tasks that you might need after the operating system has been installed include:

- Listing all available repositories:
  ```bash
  ~$ subscription-manager repos --list
  ```

- Listing all currently enabled repositories:
  ```bash
  ~$ yum repolist
  ```

- Enabling or disabling a repository:
  ```bash
  ~$ subscription-manager repos --enable repository
  ~$ subscription-manager repos --disable repository
  ```

- Searching for packages matching a specific string:
  ```bash
  ~$ yum search string
  ```

- Installing a package:
  ```bash
  ~$ yum install package_name
  ```

- Updating all packages and their dependencies:
  ```bash
  ~$ yum update
  ```

- Updating a package:
~]# yum update package_name

- Uninstalling a package and any packages that depend on it:
  
  ~]# yum remove package_name

- Listing information on all installed and available packages:
  
  ~]$ yum list all

- Listing information on all installed packages:
  
  ~]$ yum list installed

1.8. MAKING SYSTEMD SERVICES START AT BOOT TIME

Systemd is a system and service manager for Linux operating systems that introduces the concept of systemd units.

This section provides the information on how to ensure that a service is enabled or disabled at boot time. It also explains how to manage the services through the web console.

1.8.1. Enabling or disabling the services

You can determine services that are enabled or disabled at boot time already during the installation process, or you can enable or disable a service on an installed operating system.

To create the list of services enabled or disabled at boot time during the installation process, use the `services` option in the Kickstart file:

`services [--disabled=list] [--enabled=list]`

**NOTE**

The list of disabled services is processed before the list of enabled services. Therefore, if a service appears on both lists, it will be enabled. The list of the services should be given in the comma separated format. Do not include spaces in the list of services.

To enable or disable a service on an already installed operating system:

`~]# systemctl enable service_name`

`~]# systemctl disable service_name`

For further details on enabling and disabling services, see Section 3.2, “Managing system services”.

1.8.2. Managing services in the RHEL 8 web console

In the web console, select Services to manage systemd targets, services, sockets, timers and paths. There you can check their status, start or stop them, enable or disable them.
Computer security is the protection of computer systems from the theft or damage to their hardware, software, or information, as well as from disruption or misdirection of the services they provide. Ensuring computer security is therefore an essential task not only in the enterprises processing sensitive data or handling some business transactions.

Computer security includes a wide variety of features and tools. This section covers only the basic security features that you need to configure after you have installed the operating system. For detailed information on securing Red Hat Enterprise Linux, see Configuring and managing security.

1.9.1. Ensuring the firewall is enabled and running

1.9.1.1. What a firewall is and how it enhances system security

A firewall is a network security system that monitors and controls the incoming and outgoing network traffic based on predetermined security rules. A firewall typically establishes a barrier between a trusted, secure internal network and another outside network.

The firewall is provided by the firewalld service, which is automatically enabled during the installation. However, if you explicitly disabled the service, you can re-enable it, as described in Section 1.9.1.2, "Re-enabling the firewalld service".

1.9.1.2. Re-enabling the firewalld service

In case that the firewalld service is disabled after the installation, Red Hat recommends to consider re-enabling it.

You can display the current status of firewalld even as a regular user:

```bash
$ systemctl status firewalld
```

If firewalld is not enabled and running, switch to the root user, and change its status:

```bash
# systemctl start firewalld
# systemctl enable firewalld
```

For detailed information on configuring and using firewall, see Using and configuring firewalls.
1.9.1.3. Managing firewall in the RHEL 8 web console

In the web console, use the **Firewall** option under **Networking** to enable or disable the **firewalld** service.

By default, the **firewalld** service in the web console is enabled. To disable it, set **off** as shown below. Additionally, you can choose the services that you want to allow through firewall.

**Figure 1.3. Managing firewall in the RHEL 8 web console**

![Firewall in RHEL 8 web console](image)

1.9.2. Ensuring the appropriate state of SELinux

1.9.2.1. What SELinux is and how it enhances system security

**Security Enhanced Linux (SELinux)** is an additional layer of system security that determines which process can access which files, directories, and ports.

**SELinux states**

- **Enabled**
- **Disabled**

When **SELinux** is disabled, only Discretionary Access Control (DAC) rules are used.

**SELinux modes**

When **SELinux** is enabled, it can run in one of the following modes:

- **Enforcing**
- **Permissive**

Enforcing mode means that **SELinux** policies are enforced. **SELinux** denies access based on **SELinux** policy rules, and enables only the interactions that are particularly allowed. Enforcing mode is the default mode after the installation and it is also the safest **SELinux** mode.

Permissive mode means that **SELinux** policies are not enforced. **SELinux** does not deny access, but denials are logged for actions that would have been denied if running in enforcing mode. Permissive mode is the default mode during the installation. Operating in permissive mode is also useful in some specific cases, for example if you require access to the Access Vector Cache (AVC) denials when troubleshooting problems.

For further information on **SELinux**, see [Configuring and managing security](#).

1.9.3. Ensuring the Required State of SELinux
By default, **SELinux** operates in permissive mode during the installation and in enforcing mode when the installation has finished.

However, in some specific scenarios, **SELinux** might be explicitly set to permissive mode or it might even be disabled on the installed operating system. This can be set for example in the kickstart configuration.

**IMPORTANT**

Red Hat recommends to keep your system in enforcing mode.

To display the current **SELinux** mode, and to set the mode as needed:

**Ensuring the required state of SELinux**

1. Display the current **SELinux** mode in effect:

   ```bash
   ~]$ getenforce
   ```

2. If needed, switch between the **SELinux** modes.
   The switch can be either temporary or permanent. A temporary switch is not persistent across reboots, while permanent switch is.

   - To temporary switch to either enforcing or permissive mode:

     ```bash
     ~]$ setenforce Enforcing
     ~]$ setenforce Permissive
     ```

   - To permanently set the **SELinux** mode, modify the **SELINUX** variable in the `/etc/selinux/config` configuration file.
     For example, to switch **SELinux** to enforcing mode:

     ```bash
     # This file controls the state of SELinux on the system.
     # SELINUX= can take one of these three values:
     #     enforcing - SELinux security policy is enforced.
     #     permissive - SELinux prints warnings instead of enforcing.
     #     disabled - No SELinux policy is loaded.
     SELINUX=enforcing
     ```

1.9.3.1. Managing SELinux in the RHEL 8 web console

In the web console, use the **SELinux** option to turn **SELinux** enforcing policy on or off.

By default, **SELinux** enforcing policy in the web console is on, and **SELinux** operates in enforcing mode. By turning it off, you can switch **SELinux** into permissive mode. Note that such deviation from the default configuration in the `/etc/sysconfig/selinux` file is automatically reverted on the next boot.
1.9.4. Managing SELinux using RHEL System Roles

You can manage various SELinux local customizations on multiple target machines using the `selinux` role.

The following example shows how to apply the `selinux` role to set SELinux mode and reboot the managed host if required to apply the changes:

```
Example 1.2. An example playbook applying the selinux role for setting SELinux mode

- hosts: selinux-test
  vars:
    # possible values: enforcing, permissive, disabled
    selinux_state: enforcing
  tasks:
    - name: execute the role and catch errors
      block:
        - include_role:
          name: rhel-system-roles.selinux
      rescue:
        # Fail if failed for a different reason than selinux_reboot_required.
        - name: handle errors
          fail:
            msg: "role failed"
            when: not selinux_reboot_required
        - name: Unconditionally reboot the machine with all defaults
          reboot:
```

For more information on applying the `selinux` role to manage various local customizations, such as applying restorecon to portions of file system tree, or managing file contexts, SELinux booleans, logins, or ports, see System roles documentation.
1.9.5. Accessing system through SSH

1.9.5.1. What SSH-based access is and how it enhances system security

If you want to secure your communication with another computer, you can use SSH-based authentication.

Secure Shell (SSH) is a protocol which facilitates client-server communication and allows users to log in to any host system running SSH remotely. SSH secures the connection. The client transmits its authentication information to the server using encryption, and all data sent and received during a session are transferred under the encryption.

SSH enables its users to authenticate without a password. To do so, SSH uses a private-public key scheme.

For more information about SSH, see Configuring and managing security.

1.9.5.2. Configuring key-based SSH access

To be able to use SSH connection, create a pair of two keys consisting of a public and a private key.

Creating the key files and Copying them to the Server

1. Generate a public and a private key:

   ```
   ~]$ ssh-keygen
   ```

   Both keys are stored in the ~/.ssh/ directory:

   - ~/.ssh/id_rsa.pub - public key
   - ~/.ssh/id_rsa - private key

   The public key does not need to be secret. It is used to verify the private key. The private key is secret. You can choose to protect the private key with the passphrase that you specify during the key generation process. With the passphrase, authentication is even more secure, but is no longer password-less. You can avoid this using the ssh-agent command. In this case, you will enter the passphrase only once - at the beginning of a session.

2. Copy the most recently modified public key to a remote machine you want to log into:

   ```
   ~]$ ssh-copy-id USER@hostname
   ```

   As a result, you are now able to enter the system in a secure way, but without entering a password.

1.9.5.3. Disabling SSH root login

To increase system security, you can disable SSH access for the root user, which is enabled by default.

Disabling SSH root login

1. Access the /etc/ssh/sshd_config file:

   ```
   ~]$ vi /etc/ssh/sshd_config
   ```
2. Change the line that reads `#PermitRootLogin yes` to:

```
PermitRootLogin no
```

3. Restart the `sshd` service:

```
~]# systemctl restart sshd
```

**NOTE**
When using `PermitRootLogin no`, the `root` user cannot login to system directly. Alternatively, the `root` user might be allowed to login, but using a password-less authentication method, typically the key-based authentication, described in Section 1.9.5.2, "Configuring key-based SSH access". To ensure this, specify `PermitRootLogin prohibit-password` in the `/etc/ssh/sshd_config` file.

1.10. MANAGING USER ACCOUNTS

Red Hat Enterprise Linux is a multi-user operating system, which enables multiple users on different computers to access a single system installed on one machine. Every user operates under its own account, and managing user accounts thus represents a core element of Red Hat Enterprise Linux system administration.

1.10.1. The basics of managing user accounts

**Normal and System Accounts**

Normal accounts are created for users of a particular system. Such accounts can be added, removed, and modified during normal system administration.

System accounts represent a particular applications identifier on a system. Such accounts are generally added or manipulated only at software installation time, and they are not modified later.

**WARNING**

System accounts are presumed to be available locally on a system. If these accounts are configured and provided remotely, such as in the instance of an LDAP configuration, system breakage and service start failures can occur.

For system accounts, user IDs below 1000 are reserved. For normal accounts, you can use IDs starting at 1000. However, the recommended practice is to assign IDs starting at 5000. See Reserved user and group IDs for more information. The guidelines for assigning IDs can be found in the `/etc/login.defs` file:

```
# Min/max values for automatic uid selection in useradd
#
UID_MIN          1000
UID_MAX          60000
```
# System accounts

| SYS_UID_MIN | 201 |
| SYS_UID_MAX | 999 |

### What groups are and which purposes they can be used for

A group in an entity which ties together multiple user accounts for a common purpose, such as granting access to particular files.

#### 1.10.1.1. Basic command-line tools to manage user accounts and groups

The most basic tasks to manage user accounts and groups, and the appropriate command-line tools, include:

- Displaying user and group IDs:
  ```
  ~]$ id
  ```
- Creating a new user account:
  ```
  ~]$ useradd [options] user_name
  ```
- Assigning a new password to a user account belonging to username:
  ```
  ~]$ passwd user_name
  ```
- Adding a user to a group:
  ```
  ~]$ usermod -a -G group_name user_name
  ```

For detailed information on managing users and groups, see Section 4.1, “Introduction to Users and Groups”.

#### 1.10.2. System user accounts managed in the web console

With user accounts displayed in the RHEL 8 web console you can:

- Authenticate users when accessing the system.
- Set them access rights to the system.

The RHEL 8 web console displays all user accounts located in the system. Therefore, you can see at least one user account just after the first login to the web console.

Once you are logged in to the RHEL 8 web console, you can:

- Create new users accounts.
- Change their parameters.
- Lock accounts.
- Terminate the user session.

You can find the account management in the Accounts settings.
1.10.3. Adding new accounts in the web console

The following describes adding system user accounts in the RHEL 8 web console and setting administration rights to the accounts.

Procedure

1. Log in to the RHEL web console.

2. Click Accounts.

3. Click Create New Account

4. In the Full Name field, enter the full name of the user.
   The RHEL web console automatically suggests a user name from the full name and fills it in the User Name field. If you do not want to use the original naming convention consisting of the first letter of the first name and the whole surname, update the suggestion.

5. In the Password/Confirm fields, enter the password and retype it for verification that your password is correct. The color bar placed below the fields shows you security level of the entered password, which does not allow you to create a user with a weak password.

6. Click Create to save the settings and close the dialog box.

7. Select the newly created account.

8. Select Server Administrator in the Roles item.
Now you can see the new account in the **Accounts** settings and you can use the credentials to connect to the system.

### 1.11. DUMPING THE CRASHED KERNEL USING THE KDUMP MECHANISM

This section provides an introduction to the kernel crash dump mechanism, also called *kdump*, and briefly explains what *kdump* is used for in Section 1.11.1, “What *kdump* is and which tasks it can be used for”.

Activation of the *kdump* service is a part of the installation process, as described in Installing Red Hat Enterprise Linux 8. This section summarizes how to manually enable the *kdump* service if it is disabled after the installation in Section 1.11.2, “Ensuring that *kdump* is installed and enabled after the installation process”.

You can also use the web console to configure *kdump*. See Section 1.11.3, “Configuring *kdump* in the RHEL 8 web console” for more information.

#### 1.11.1. What *kdump* is and which tasks it can be used for

In case of a system crash, you can use the kernel crash dump mechanism called *kdump* that enables you to save the content of the system’s memory for later analysis. The *kdump* mechanism relies on the kexec system call, which can be used to boot a Linux kernel from the context of another kernel, bypass BIOS, and preserve the contents of the first kernel’s memory that would otherwise be lost.

When kernel crash occurs, *kdump* uses kexec to boot into a second kernel, a capture kernel, which resides in a reserved part of the system memory that is inaccessible to the first kernel. The second kernel captures the contents of the crashed kernel’s memory, a crash dump, and saves it.

For more detailed information about *kdump*, see Managing, monitoring and updating the kernel.

#### 1.11.2. Ensuring that *kdump* is installed and enabled after the installation process

To ensure that *kdump* is installed, and to configure it:

**Checking whether *kdump* is installed and Configuring *kdump***
1. To check whether **kdump** is installed on your system:

   ```
   ~$ rpm -q kexec-tools
   ```

2. If not installed, to install **kdump**, enter as the **root** user:

   ```
   ~# yum install kexec-tools
   ```

3. To configure **kdump**:

   Use either the command line or graphical user interface as described in **Managing, monitoring and updating the kernel**.

   If you need to install the graphical configuration tool:

   ```
   ~# yum install system-config-kdump
   ```

### 1.11.3. Configuring kdump in the RHEL 8 web console

In the web console, select **Kernel dump configuration** to verify:

- the **kdump** status
- the amount of memory reserved for **kdump**
- the location of the crash dump files

![Figure 1.5. Configuring kdump in the RHEL 8 web console](image)

### 1.11.4. Configuring kdump using RHEL System Roles

You can configure **kdump** on multiple target machines using the **kdump** role.
The following example shows how to apply the `kdump` role to set the location of the crash dump files:

```yaml
---
- hosts: kdump-test
  vars:
    kdump_path: /var/crash
  roles:
    - rhel-system-roles.kdump
```

For more information on applying the `kdump` role to manage various `kdump` configurations, see System roles documentation.

### 1.12. PERFORMING SYSTEM RESCUE AND CREATING SYSTEM BACKUP WITH REAR

When a software or hardware failure breaks the operating system, you need a mechanism to rescue the system. It is also useful to have the system backup saved. Red Hat recommends using the Relax-and-Recover (ReaR) tool to fulfil both these needs.

#### 1.12.1. What ReaR is and which tasks it can be used for

ReaR is a disaster recovery and system migration utility which enables you to create the complete rescue system. By default, this rescue system restores only the storage layout and the boot loader, but not the actual user and system files.

Additionally, certain backup software enables you to integrate ReaR for disaster recovery.

ReaR enables to perform the following tasks:

- Booting a rescue system on the new hardware
- Replicating the original storage layout
- Restoring user and system files

#### 1.12.2. Quickstart to installation and configuration of ReaR

To install ReaR, enter as the `root` user:
1.12.3. Quickstart to creation of the rescue system with ReaR

To create the rescue system, perform the following command as the root user:

```bash
~$ rear mkrescue
```

1.12.4. Quickstart to configuration of ReaR with the backup software

ReaR contains a fully-integrated built-in, or internal, backup method called NETFS.

To make ReaR use its internal backup method, add these lines to the `/etc/rear/local.conf` file:

```text
BACKUP=NETFS
BACKUP_URL=backup location
```

You can also configure ReaR to keep the previous backup archives when the new ones are created by adding the following line to `/etc/rear/local.conf`:

```text
NETFS_KEEP_OLD_BACKUP_COPY=y
```

To make the backups incremental, meaning that only the changed files are backed up on each run, add this line to `/etc/rear/local.conf`:

```text
BACKUP_TYPE=incremental
```

1.13. USING THE LOG FILES TO TROUBLESHOOT PROBLEMS

When troubleshooting a problem, you may appreciate the log files that contain different information and messages about the operating system. The logging system in Red Hat Enterprise Linux is based on the built-in syslog protocol. Particular programs use this system to record events and organize them into log files, which are useful when auditing the operating system and troubleshooting various problems.

1.13.1. Services handling the syslog messages

The syslog messages are handled by two services:

- The `systemd-journald` daemon
- The `rsyslog` service

The `systemd-journald` daemon collects messages from various sources and forwards them to the `rsyslog` service for further processing. The sources from which the messages are collected are:

- Kernel
- Early stages of the boot process
- Standard output and error of daemons as they start up and run
1.13.2. Subdirectories storing the syslog messages

The syslog messages are stored in various subdirectories under the /var/log directory according to what kind of messages and logs they contain:

- **var/log/messages** - all syslog messages except those mentioned below
- **var/log/secure** - security and authentication-related messages and errors
- **var/log/maillog** - mail server-related messages and errors
- **var/log/cron** - log files related to periodically executed tasks
- **var/log/boot.log** - log files related to system startup

1.13.2.1. Managing the log files in the RHEL 8 web console

in the web console, use the **Logs** option if you want to inspect the log files.

1.14. ACCESSING RED HAT SUPPORT

To obtain support from Red Hat, use the Red Hat Customer Portal, which provides access to everything available with your subscription.

This section describes:

- Using the **SOS report** to troubleshoot problems, in Section 1.14.2, “Using the SOS report to troubleshoot problems”


By using the Red Hat Customer Portal you can:

- Open a new support case
- Initiate a live chat with a Red Hat expert
1.14.2. Using the SOS report to troubleshoot problems

The **SOS report** collects configuration details, system information and diagnostic information from a Red Hat Enterprise Linux system. Attach the report when you open a support case.

Note that the **SOS report** is provided in the **sos** package, which is not installed with the default minimal installation of Red Hat Enterprise Linux.

To install the **sos** package:

```
~]# yum install sos
```

To generate an **SOS report**:

```
~]# sosreport
```

To attach the **sos report** to your support case, see the Red Hat Knowledgebase article [How can I attach a file to a Red Hat support case?](https://access.redhat.com). Note that you will be prompted to enter the number of the support case, when attaching the **sos report**.

For more information on **SOS report**, see the Red Hat Knowledgebase article [What is a sosreport and how to create one in Red Hat Enterprise Linux 4.6 and later?](https://access.redhat.com).
CHAPTER 2. INSTALLING SOFTWARE WITH YUM

2.1. INTRODUCTION TO INSTALLING SOFTWARE ON RED HAT ENTERPRISE LINUX 8

On Red Hat Enterprise Linux 8, installing software is ensured by the new version of the YUM tool, which is based on the DNF technology (YUM v4).

YUM v4 has the following advantages over the previous YUM v3 used on Red Hat Enterprise Linux 7:

- Increased performance
- New features available, most significantly the support for managing the modular content
- Well-designed stable API for integration with tooling

For detailed information about differences between the new YUM tool and the previous version YUM v3 from Red Hat Enterprise Linux 7, see Changes in DNF CLI compared to YUM.

**NOTE**

Note, that upstream calls this tool DNF. As a result, some output returned by the new YUM tool in Red Hat Enterprise Linux 8 mentions DNF, and upstream documentation identifies the technology as DNF.

For installing software, you can use the `yum` command and its particular options in the same way as on Red Hat Enterprise Linux 7.

Selected yum plug-ins and utilities have been ported to the new DNF back end, and can be installed under the same names as in Red Hat Enterprise Linux 7. They also provide compatibility symlinks, so the binaries, configuration files and directories can be found in usual locations.

Note that the legacy Python API provided by YUM v3 is no longer available. Users are advised to migrate their plug-ins and scripts to the new API provided by YUM v4 (DNF Python API), which is stable and fully supported. The DNF Python API is available here.

The Libdnf and Hawkey APIs (both C and Python) are unstable, and will likely change during RHEL 8 life cycle.

### 2.1.1. Application streams

Red Hat Enterprise Linux 8.0 introduces the concept of Application Streams. Multiple versions of user space components are now delivered and updated more frequently than the core operating system packages. This provides greater flexibility to customize Red Hat Enterprise Linux without impacting the underlying stability of the platform or specific deployments.

Components made available as Application Streams can be packaged as modules or RPM packages and are delivered through the AppStream repository in RHEL 8. Each Application Stream component has a given life cycle. For details, see Red Hat Enterprise Linux Life Cycle.

Modules are collections of packages representing a logical unit: an application, a language stack, a database, or a set of tools. These packages are built, tested, and released together.

Module streams represent versions of the Application Stream components. For example, two streams
(versions) of the PostgreSQL database server are available in the postgresql module: PostgreSQL 10 (the default stream) and PostgreSQL 9.6. Only one module stream can be installed on the system. Different versions can be used in separate containers.

Detailed module commands are described in the Installing, managing, and removing user space components document. For a list of modules available in AppStream, see the Package manifest.

2.2. INTRODUCTION TO YUM FUNCTIONALITY

**yum** is the Red Hat package manager that is able to query for information about available packages, fetch packages from repositories, install and uninstall them, and update an entire system to the latest available version. Yum performs automatic dependency resolution when updating, installing, or removing packages, and thus is able to automatically determine, fetch, and install all available dependent packages.

**yum** can be configured with new, additional repositories, or package sources, and also provides many plug-ins which enhance and extend its capabilities. Yum enables easy and simple package management.

**IMPORTANT**

**yum** provides secure package management by enabling Gnu Privacy Guard (GPG), also known as GnuPG, signature verification on GPG–signed packages to be turned on for all package repositories (package sources), or for individual repositories.

You can also use **yum** to set up your own repositories with RPM packages for download and installation on other machines. When possible, yum uses parallel download of multiple packages and metadata to speed up downloading.

**NOTE**

You must have superuser privileges in order to use **yum** to install, update or remove packages on your system. All examples here assume that you have already obtained superuser privileges by using either the su or sudo command.

2.3. USING YUM FOR PARTICULAR TASKS

This section describes how to use **yum** to achieve particular tasks.

2.3.1. Checking for updates and updating packages

**yum** enables you to check if your system has any updates waiting to be applied. You can list packages that need to be updated and update them as a whole, or you can update a selected individual package.

2.3.1.1. Checking for updates

To see which installed packages on your system have updates available, use the following command:

```
yum check-update
```

The command shows the list of packages and their dependencies that have an update available. The output for each package consists of:

- the name of the package
the CPU architecture the package was built for
- the version of the updated package to be installed
- the release of the updated package
- a build version, added as part of a z-stream update
- the repository in which the updated package is located.

2.3.1.1.1. Updating packages

You can choose to update a single package, multiple packages, or all packages at once. If any dependencies of the package or packages you update have updates available themselves, then they are updated too.

2.3.1.1.1.1. Updating a single package

To update a single package, run the following command as root:

```shell
yum update package_name
```

**IMPORTANT**

`yum` always installs a new kernel regardless of whether you are using the `yum update` or `yum install` command to apply kernel updates.

When using RPM, on the other hand, it is important to use the `rpm -i kernel` command which installs a new kernel instead of `rpm -u kernel` which replaces the current kernel.

2.3.1.1.1.2. Updating a package group

To update a package group, type as root:

```shell
yum group update group_name
```

Here, replace `group_name` with a name of the package group you want to update. For more information on package groups, see Section 2.3.3, “Working with package groups”.

2.3.1.1.2. Updating all packages and their dependencies

To update all packages and their dependencies, use the `yum update` command without any arguments:

```shell
yum update
```

2.3.1.1.3. Updating security-related packages

If packages have security updates available, you can update only these packages to their latest versions. Type as root

```shell
yum update --security
```

You can also update packages only to versions containing the latest security updates. Type as root.
2.3.2. Working with packages

Yum enables you to perform a complete set of operations with software packages, including searching for packages, viewing information about them, installing and removing.

2.3.2.1. Searching packages

You can search all package names, descriptions and summaries by using the following command:

```sh
yum search term...
```

Replace `term` with a package name you want to search.

The `yum search` command is useful for searching for packages you do not know the name of, but for which you know a related term. Note that by default, `yum search` returns matches in package name and summary, which makes the search faster. Use the `yum search --all` command for a more exhaustive but slower search, which also includes package descriptions.

2.3.2.1.1. Filtering the Results

All of yum's list commands allow you to filter the results by appending one or more `glob expressions` as arguments. Global expressions are normal strings of characters which contain one or more of the wildcard characters `*` (which expands to match any character subset) and `?` (which expands to match any single character).

Be careful to escape the global expressions when passing them as arguments to a `yum` command, otherwise the Bash shell will interpret these expressions as `pathname expansions`, and potentially pass all files in the current directory that match the global expressions to `yum`. To make sure the global expressions are passed to `yum` as intended, use one of the following methods:

- escape the wildcard characters by preceding them with a backslash character
- double-quote or single-quote the entire global expression.

2.3.2.2. Listing packages

To list information on all installed and available packages type the following at a shell prompt:

```sh
yum list --all
```

To list installed and available packages that match inserted global expressions use the following command:

```sh
yum list glob_expression...
```

To list all packages installed on your system use the `installed` keyword.

```sh
yum list --installed glob_expression...
```

To list all packages in all enabled repositories that are available to install, use the command in the following form:
yum list --available glob_expression...

2.3.2.2.1. Listing repositories

To list the repository ID, name, and number of packages for each enabled repository on your system, use the following command:

yum repolist

To list more information about these repositories, use the repoinfo command. With this command, information including the file name, overall size, date of the last update, and base URL are displayed for each listed repository.

yum repoinfo

To list both enabled and disabled repositories use the following command. A status column is added to the output list to show which of the repositories are enabled.

yum repolist --all

By passing disabled as a first argument, you can reduce the command output to disabled repositories. For further specification you can pass the ID or name of repositories or related glob_expressions as arguments. Note that if there is an exact match between the repository ID or name and the inserted argument, this repository is listed even if it does not pass the enabled or disabled filter.

2.3.2.2.2. Displaying package information

To display information about one or more packages, use the following command (global expressions are valid here as well):

yum info package_name...

Replace package_name with the name of the package.

2.3.2.2.3. Installing packages

To install a single package and all of its non-installed dependencies, enter a command in the following form as root:

yum install package_name

You can also install multiple packages simultaneously by appending their names as arguments. To do so, type as root:

yum install package_name package_name...

If you are installing packages on a multilib system, such as an AMD64 or Intel 64 machine, you can specify the architecture of the package (as long as it is available in an enabled repository) by appending .arch to the package name:

yum install package_name.arch
You can use global expressions to quickly install multiple similarly named packages. Execute as root:

```
yum install glob_expression...
```

In addition to package names and global expressions, you can also provide file names to `yum install`. If you know the name of the binary you want to install, but not its package name, you can give `yum install` the path name. As root, type:

```
yum install /usr/sbin/named
```

Yum then searches through its package lists, finds the package which provides `/usr/sbin/named`, if any, and prompts you as to whether you want to install it.

As you can see in the above examples, the `yum install` command does not require strictly defined arguments. It can process various formats of package names and global expressions, which makes installation easier for users. On the other hand, it takes some time until `yum` parses the input correctly, especially if you specify a large number of packages. To optimize the package search, you can use the following commands to explicitly define how to parse the arguments:

```
yum install-n name
yum install-na name.architecture
yum install-nevra name-epoch:version-release.architecture
```

With `install-n`, yum interprets `name` as the exact name of the package. The `install-na` command tells yum that the subsequent argument contains the package name and architecture divided by the dot character. With `install-nevra`, yum will expect an argument in the form `name-epoch:version-release.architecture`. Similarly, you can use `yum remove-n`, `yum remove-na`, and `yum remove-nevra` when searching for packages to be removed.

**NOTE**

If you know you want to install the package that contains the `named` binary, but you do not know in which `bin/` or `sbin/` directory the file is installed, use the `yum provides` command with a global expression.

```
yum provides "/file_name"
```

is a useful way to find the packages that contain `file_name`.

To install a previously-downloaded package from a local directory on your system, use the following command:

```
yum install path
```

Replace `path` with the path to the package you want to install.

Alternatively, you can use also the `yum localinstall` command to install a previously -nloaded package from a local directory.

2.3.2.2.4. Removing packages

To uninstall a particular package, as well as any packages that depend on it, run the following command as root:
yum remove package_name...

To remove multiple packages at once by adding more package names to the command.

Similar to install, remove can take these arguments:

- package names
- global expressions
- file lists
- package provides

WARNING

yum is not able to remove a package without also removing packages which depend on it.

2.3.3. Working with package groups

A package group is a collection of packages that serve a common purpose, for instance System Tools or Sound and Video. Installing a package group pulls a set of dependent packages, saving time considerably. The yum groups command is a top-level command that covers all the operations that act on package groups in yum.

2.3.3.1. Listing package groups

The summary option is used to view the number of:

- installed groups
- available groups
- available environment groups
- installed and available language groups

yum groups summary

To list all package groups from yum repositories add the list option. You can filter the command output by group names.

yum group list glob_expression...

Several optional arguments can be passed to this command, including --hidden to list also groups not marked as user visible, and -v to list group IDs. You can also add --installed, or --available options to reduce the command output to a specific group type.

To list mandatory and optional packages contained in a particular group, use the following command:
**yum group info glob_expression**...

**NOTE**

A package group can be marked with @. When using **yum group list, info, install, or remove**, pass **@group_name** to specify a package group or an environmental group.

### 2.3.3.2. Installing a package group

You can install a package group by passing its full group name, without the groupid part, to the **group install** command. As **root**, type:

```
yum group install group_name
```

You can also install by groupid. As **root**, execute the following command:

```
yum group install groupid
```

You can pass the groupid or quoted group name to the **install** command if you prepend it with an @ symbol, which tells **yum** that you want to perform **group install**. As **root**, type:

```
yum install @group
```

Replace **group** with the groupid or quoted group name. The same logic applies to environmental groups:

```
yum install @group
```

### 2.3.3.3. Removing a package group

You can remove a package group using syntax similar to the **install** syntax, with use of either name of the package group or its id. As **root**, type:

```
yum group remove group_name
```

```
yum group remove groupid
```

Also, you can pass the groupid or quoted name to the **remove** command if you prepend it with an @ symbol, which tells **yum** that you want to perform **group remove**. As **root**, type:

```
yum remove @group
```

Replace **group** with the groupid or quoted group name. Similarly, you can replace an environmental group:

```
yum remove @group
```

### 2.4. WORKING WITH TRANSACTION HISTORY

The **yum history** command enables users to review information about a timeline of yum transactions,
the dates and times they occurred, the number of packages affected, whether these transactions
succeeded or were aborted, and if the RPM database was changed between transactions. Additionally,
this command can be used to undo or redo certain transactions. All history data is stored in the history
DB in the /var/lib/yum/history/ directory.

2.4.1. Listing transactions

To display a list of the twenty most recent transactions, as root, either run yum history with no
additional arguments, or type the following at a shell prompt:

```bash
yum history list
```

To examine a particular transaction or transactions in more detail, run the following command as root:

```bash
yum history info id...
```

The `id` argument here stands for the ID of the transaction. This argument is optional and when you omit
it, yum automatically uses the last transaction.

2.4.2. Reverting and repeating transactions

Apart from reviewing the transaction history, the `yum history` command provides means to revert or
repeat a selected transaction. To revert a transaction, type the following at a shell prompt as root:

```bash
yum history undo id
```

To repeat a particular transaction, as root, run the following command:

```bash
yum history redo id
```

Both commands also accept the `last` keyword to undo or repeat the latest transaction.

Note that both `yum history undo` and `yum history redo` commands only revert or repeat the steps
that were performed during a transaction. If the transaction installed a new package, the `yum history
undo` command will uninstall it, and if the transaction uninstalled a package the command will again
install it. This command also attempts to downgrade all updated packages to their previous version, if
these older packages are still available.

2.5. CONFIGURING YUM AND YUM REPOSITORIES

The configuration information for yum and related utilities is located in the /etc/yum.conf file. This file
contains one mandatory [main] section, which enables you to set yum options that have global effect,
and can also contain one or more [repository] sections, which allow you to set repository-specific
options. However, it is recommended to define individual repositories in new or existing .repo files in the
/etc/yum.repos.d/ directory. The values you define in individual [repository] sections of the
/etc/yum.conf file override values set in the [main] section.

This section shows how to:

- set global yum options by editing the [main] section of the /etc/yum.conf configuration file;
- set options for individual repositories by editing the [repository] sections in /etc/yum.conf and
  .repo files in the /etc/yum.repos.d/ directory;
2.5.1. Viewing the current configuration

To display the current values of global yum options (that is, the options specified in the [main] section of the /etc/yum.conf file), execute the yum-config-manager command with the following command-line option:

```
yum-config-manager --dump
```

2.5.2. Setting [main] options

The /etc/yum.conf configuration file contains exactly one [main] section, and while some of the key-value pairs in this section affect how yum operates, others affect how yum treats repositories.

You can add many additional options under the [main] section heading in /etc/yum.conf.

For a complete list of available [main] options, see the [main] OPTIONS section of the yum.conf(5) manual page.

2.5.3. Setting [repository] options

The [repository] sections, where repository is a unique repository ID such as my_personal_repo (spaces are not permitted), allow you to define individual yum repositories. To avoid conflicts, do not use names used by Red Hat repositories for custom repositories.

For a complete list of available [repository] options, see the [repository] OPTIONS section of the yum.conf(5) manual page.

2.5.4. Adding, enabling, and disabling a yum repository

Section 2.5.3, “Setting [repository] options” describes various options you can use to define a yum repository. This section explains how to add, enable, and disable a repository by using the yum-config-manager command.

2.5.4.1. Adding a yum repository

To define a new repository, you can either add a [repository] section to the /etc/yum.conf file, or to a .repo file in the /etc/yum.repos.d/ directory. All files with the .repo file extension in this directory are read by yum, and it is recommended to define your repositories here instead of in /etc/yum.conf.

---

**WARNING**

Obtaining and installing software packages from unverified or untrusted software sources other than Red Hat’s certificate-based Content Delivery Network (CDN) constitutes a potential security risk, and could lead to security, stability, compatibility, and maintainability issues.
Yum repositories commonly provide their own .repo file. To add such a repository to your system and enable it, run the following command as root:

```
yum-config-manager --add-repo repository_url
```

Here repository_url is a link to the .repo file.

### 2.5.4.2. Enabling a yum repository

To enable a particular repository or repositories, type the following at a shell prompt as root:

```
yum-config-manager --enable repository...
```

Here repository is the unique repository ID (use yum repolist all to list available repository IDs).

### Disabling a yum repository

To disable a yum repository, run the following command as root:

```
yum-config-manager --disable repository...
```

Here repository is the unique repository ID (use yum repolist all to list available repository IDs).

### 2.6. USING YUM PLUG-INS

Yum provides plug-ins that extend and enhance its operations. Certain plug-ins are installed by default. Yum always informs you which plug-ins, if any, are loaded and active whenever you call any yum command.

#### 2.6.1. Enabling, configuring, and disabling yum plug-ins

To enable yum plug-ins, ensure that a line beginning with plugins= is present in the [main] section of /etc/yum.conf, and that its value is 1:

```
plugins=1
```

You can disable all plug-ins by changing this line to plugins=0.

**IMPORTANT**

Disabling all plug-ins is not advised because certain plug-ins provide important yum services. In particular, the product-id and subscription-manager plug-ins provide support for the certificate-based Content Delivery Network (CDN). Disabling plug-ins globally is provided as a convenience option, and is generally only recommended when diagnosing a potential problem with yum.

Every installed plug-in has its own configuration file in the /etc/dnf/plugins/ directory. You can set plug-in specific options in these files.

Similar to the /etc/yum.conf file, the plug-in configuration files always contain a [main] section where the enabled= option controls whether the plug-in is enabled when you run yum commands. If this option is missing, you can add it manually to the file.
If youdisable all plug-ins by setting `enabled=0` in `/etc/yum.conf`, then all plug-ins are disabled regardless of whether they are enabled in their individual configuration files.

If you want to disable all yum plug-ins for a single yum command, use the `--noplugins` option.

If you want to disable one or more yum plug-ins for a single yum command, add the `--disableplugin=plugin_name` option to the command.

2.7. ADDITIONAL RESOURCES

The following sources of information provide additional resources regarding YUM.

2.7.1. Installed Documentation

- `yum(8)` – The manual page for the yum command-line utility provides a complete list of supported options and commands.

- `yum.conf(5)` – The manual page named `yum.conf` documents available yum configuration options.

2.7.2. Online Documentation

- Red Hat Customer Portal Labs – The Red Hat Customer Portal Labs includes a “Yum Repository Configuration Helper”.
CHAPTER 3. MANAGING SERVICES WITH SYSTEMD

3.1. INTRODUCTION TO SYSTEMD

Systemd is a system and service manager for Linux operating systems. It is designed to be backwards compatible with SysV init scripts, and provides a number of features such as parallel startup of system services at boot time, on-demand activation of daemons, or dependency-based service control logic. Starting with Red Hat Enterprise Linux 7, systemd replaced Upstart as the default init system.

Systemd introduces the concept of systemd units. These units are represented by unit configuration files located in one of the directories listed in the following table.

Table 3.1. Systemd unit files locations

<table>
<thead>
<tr>
<th>Directory</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/lib/systemd/system/</td>
<td>Systemd unit files distributed with installed RPM packages.</td>
</tr>
<tr>
<td>/run/systemd/system/</td>
<td>Systemd unit files created at run time. This directory takes precedence over the directory with installed service unit files.</td>
</tr>
<tr>
<td>/etc/systemd/system/</td>
<td>Systemd unit files created by systemctl enable as well as unit files added for extending a service. This directory takes precedence over the directory with runtime unit files.</td>
</tr>
</tbody>
</table>

The units encapsulate information about:

- System services
- Listening sockets
- Other objects that are relevant to the init system

For a complete list of available systemd unit types, see the following table.

Table 3.2. Available systemd unit types

<table>
<thead>
<tr>
<th>Unit Type</th>
<th>File Extension</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Service unit</td>
<td>.service</td>
<td>A system service.</td>
</tr>
<tr>
<td>Target unit</td>
<td>.target</td>
<td>A group of systemd units.</td>
</tr>
<tr>
<td>Automount unit</td>
<td>.automount</td>
<td>A file system automount point.</td>
</tr>
<tr>
<td>Device unit</td>
<td>.device</td>
<td>A device file recognized by the kernel.</td>
</tr>
</tbody>
</table>
Overriding the default systemd configuration using system.conf
The default configuration of systemd is defined during the compilation and it can be found in the systemd configuration file at /etc/systemd/system.conf. Use this file if you want to deviate from those defaults and override selected default values for systemd units globally.

For example, to override the default value of the timeout limit, which is set to 90 seconds, use the DefaultTimeoutStartSec parameter to input the required value in seconds.

```
DefaultTimeoutStartSec=required value
```

For further information, see Example 3.20, “Changing the timeout limit”.

3.1.1. Main features

The systemd system and service manager provides the following main features:

- **Socket-based activation** – At boot time, systemd creates listening sockets for all system services that support this type of activation, and passes the sockets to these services as soon as they are started. This not only allows systemd to start services in parallel, but also makes it possible to restart a service without losing any message sent to it while it is unavailable: the corresponding socket remains accessible and all messages are queued. Systemd uses socket units for socket-based activation.

- **Bus-based activation** – System services that use D-Bus for inter-process communication can be started on-demand the first time a client application attempts to communicate with them. Systemd uses D-Bus service files for bus-based activation.

- **Device-based activation** – System services that support device-based activation can be started on-demand when a particular type of hardware is plugged in or becomes available. Systemd uses device units for device-based activation.
- **Path-based activation** — System services that support path-based activation can be started on-demand when a particular file or directory changes its state. **Systemd** uses *path units* for path-based activation.

- **Mount and automount point management** — **Systemd** monitors and manages mount and automount points. **Systemd** uses *mount units* for mount points and *automount units* for automount points.

- **Aggressive parallelization** — Because of the use of socket-based activation, **systemd** can start system services in parallel as soon as all listening sockets are in place. In combination with system services that support on-demand activation, parallel activation significantly reduces the time required to boot the system.

- **Transactional unit activation logic** — Before activating or deactivating a unit, **systemd** calculates its dependencies, creates a temporary transaction, and verifies that this transaction is consistent. If a transaction is inconsistent, **systemd** automatically attempts to correct it and remove non-essential jobs from it before reporting an error.

- **Backwards compatibility with SysV init** — **Systemd** supports SysV init scripts as described in the *Linux Standard Base Core Specification*, which eases the upgrade path to systemd service units.

### 3.1.2. Compatibility changes

The systemd system and service manager is designed to be mostly compatible with SysV init and Upstart. The following are the most notable compatibility changes with regards to Red Hat Enterprise Linux 6 system that used SysV init:

- **Systemd** has only limited support for runlevels. It provides a number of target units that can be directly mapped to these runlevels and for compatibility reasons, it is also distributed with the earlier *runlevel* command. Not all systemd targets can be directly mapped to runlevels, however, and as a consequence, this command might return **N** to indicate an unknown runlevel. It is recommended that you avoid using the *runlevel* command if possible. For more information about systemd targets and their comparison with runlevels, see [Section 3.3, “Working with systemd targets”](#).

- The *systemctl* utility does not support custom commands. In addition to standard commands such as *start*, *stop*, and *status*, authors of SysV init scripts could implement support for any number of arbitrary commands in order to provide additional functionality. For example, the init script for *iptables* could be executed with the *panic* command, which immediately enabled panic mode and reconfigured the system to start dropping all incoming and outgoing packets. This is not supported in **systemd** and the *systemctl* only accepts documented commands. For more information about the *systemctl* utility and its comparison with the earlier *service* utility, see [Table 3.3, “Comparison of the service utility with systemctl”](#).

- **systemctl** utility does not communicate with services that have not been started by **systemd**. When **systemd** starts a system service, it stores the ID of its main process in order to keep track of it. The **systemctl** utility then uses this PID to query and manage the service. Consequently, if a user starts a particular daemon directly on the command line, **systemctl** is unable to determine its current status or stop it.

- **Systemd** stops only running services. Previously, when the shutdown sequence was initiated, Red Hat Enterprise Linux 6 and earlier releases of the system used symbolic links located in the `/etc/rc0.d/` directory to stop all available system services regardless of their status. With **systemd**, only running services are stopped on shutdown.
System services are unable to read from the standard input stream. When systemd starts a service, it connects its standard input to /dev/null to prevent any interaction with the user.

System services do not inherit any context (such as the HOME and PATH environment variables) from the invoking user and their session. Each service runs in a clean execution context.

When loading a SysV init script, systemd reads dependency information encoded in the Linux Standard Base (LSB) header and interprets it at run time.

All operations on service units are subject to a default timeout of 5 minutes to prevent a malfunctioning service from freezing the system. This value is hardcoded for services that are generated from initscripts and cannot be changed. However, individual configuration files can be used to specify a longer timeout value per service, see Example 3.20, “Changing the timeout limit”.

For a detailed list of compatibility changes introduced with systemd, see the Migration Planning Guide for Red Hat Enterprise Linux 7.

3.2. MANAGING SYSTEM SERVICES

Previous versions of Red Hat Enterprise Linux, which were distributed with SysV init or Upstart, used init scripts located in the /etc/rc.d/init.d/ directory. These init scripts were typically written in Bash, and allowed the system administrator to control the state of services and daemons in their system. Starting with Red Hat Enterprise Linux 7, these init scripts have been replaced with service units.

Service units end with the .service file extension and serve a similar purpose as init scripts. To view, start, stop, restart, enable, or disable system services, use the systemctl command as described in Comparison of the service utility with systemctl, Comparison of the chkconfig utility with systemctl, and further in this section. The service and chkconfig commands are still available in the system and work as expected, but are only included for compatibility reasons and should be avoided.

Table 3.3. Comparison of the service utility with systemctl

<table>
<thead>
<tr>
<th>service</th>
<th>systemctl</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>service name start</td>
<td>systemctl start name.service</td>
<td>Starts a service.</td>
</tr>
<tr>
<td>service name stop</td>
<td>systemctl stop name.service</td>
<td>Stops a service.</td>
</tr>
<tr>
<td>service name restart</td>
<td>systemctl restart name.service</td>
<td>Restarts a service.</td>
</tr>
<tr>
<td>service name condrestart</td>
<td>systemctl try-restart name.service</td>
<td>Restarts a service only if it is running.</td>
</tr>
<tr>
<td>service name reload</td>
<td>systemctl reload name.service</td>
<td>Reloads configuration.</td>
</tr>
<tr>
<td>service</td>
<td>systemctl</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>----------------------------------------</td>
<td>---------------------------------------</td>
</tr>
<tr>
<td>service name status</td>
<td>systemctl status name.service</td>
<td>Checks if a service is running.</td>
</tr>
<tr>
<td></td>
<td>systemctl is-active name.service</td>
<td></td>
</tr>
<tr>
<td>service --status-all</td>
<td>systemctl list-units --type service --all</td>
<td>Displays the status of all services.</td>
</tr>
</tbody>
</table>

Table 3.4. Comparison of the chkconfig utility with systemctl

<table>
<thead>
<tr>
<th>chkconfig</th>
<th>systemctl</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>chkconfig name on</td>
<td>systemctl enable name.service</td>
<td>Enables a service.</td>
</tr>
<tr>
<td>chkconfig name off</td>
<td>systemctl disable name.service</td>
<td>Disables a service.</td>
</tr>
<tr>
<td>chkconfig --list name</td>
<td>systemctl status name.service</td>
<td>Checks if a service is enabled.</td>
</tr>
<tr>
<td></td>
<td>systemctl is-enabled name.service</td>
<td></td>
</tr>
<tr>
<td>chkconfig --list</td>
<td>systemctl list-unit-files --type service</td>
<td>Lists all services and checks if they are enabled.</td>
</tr>
<tr>
<td>chkconfig --list</td>
<td>systemctl list-dependencies --after</td>
<td>Lists services that are ordered to start before the specified unit.</td>
</tr>
<tr>
<td></td>
<td>systemctl list-dependencies --before</td>
<td>Lists services that are ordered to start after the specified unit.</td>
</tr>
</tbody>
</table>

Specifying service units
For clarity, all command examples in the rest of this section use full unit names with the .service file extension, for example:

```
~# systemctl stop nfs-server.service
```

However, the file extension can be omitted, in which case the systemctl utility assumes the argument is a service unit. The following command is equivalent to the one above:

```
~# systemctl stop nfs-server
```

Additionally, some units have alias names. Those names can have shorter names than units, which can be used instead of the actual unit names. To find all aliases that can be used for a particular unit, use:
Behavior of systemctl in a chroot environment
If you change the root directory using the chroot command, most systemctl commands refuse to perform any action. The reason for this is that the systemd process and the user that used the chroot command do not have the same view of the filesystem. This happens, for example, when systemctl is invoked from a kickstart file.

The exception to this are unit file commands such as the systemctl enable and systemctl disable commands. These commands do not need a running system and do not affect running processes, but they do affect unit files. Therefore, you can run these commands even in chroot environment. For example, to enable the httpd service on a system under the /srv/website1/ directory:

```bash
~]# chroot /srv/website1
~]# systemctl enable httpd.service
Created symlink /etc/systemd/system/multi-user.target.wants/httpd.service, pointing to /usr/lib/systemd/system/httpd.service.
```

3.2.1. Listing services
To list all currently loaded service units, type the following at a shell prompt:

```bash
systemctl list-units --type service
```

For each service unit file, this command displays its full name (UNIT) followed by a note whether the unit file has been loaded (LOAD), its high-level (ACTIVE) and low-level (SUB) unit file activation state, and a short description (DESCRIPTION).

By default, the systemctl list-units command displays only active units. If you want to list all loaded units regardless of their state, run this command with the --all or -a command line option:

```bash
systemctl list-units --type service --all
```

You can also list all available service units to see if they are enabled. To do so, type:

```bash
systemctl list-unit-files --type service
```

For each service unit, this command displays its full name (UNIT FILE) followed by information whether the service unit is enabled or not (STATE). For information on how to determine the status of individual service units, see Displaying service status.

Example 3.1. Listing services
To list all currently loaded service units, run the following command:

```bash
~]$ systemctl list-units --type service
UNIT                          LOAD   ACTIVE SUB     DESCRIPTION
abrt-ccpp.service             loaded active exited Install ABRT coredump hook
abrt-oops.service            loaded active running ABRT kernel log watcher
abrt-vmcore.service          loaded active exited  Harvest vmcores for ABRT
abrt-xorg.service            loaded active running ABRT Xorg log watcher
abrtd.service                loaded active running ABRT Automated Bug Reporting Tool
...
systemd-vconsole-setup.service loaded active exited  Setup Virtual Console
tog-pegasus.service loaded active running OpenPegasus CIM Server

LOAD = Reflects whether the unit definition was properly loaded.
ACTIVE = The high-level unit activation state, i.e. generalization of SUB.
SUB = The low-level unit activation state, values depend on unit type.

46 loaded units listed. Pass --all to see loaded but inactive units, too.
To show all installed unit files use ‘systemctl list-unit-files’

To list all installed service unit files to determine if they are enabled, type:

```bash
~$ systemctl list-unit-files --type service
UNIT FILE                                   STATE
abrt-ccpp.service                           enabled
abrt-oops.service                           enabled
abrt-vmcore.service                         enabled
abrt-xorg.service                           enabled
abrdtd.service                               enabled
...
wpa_supplicant.service                      disabled
ypbind.service                              disabled
```

208 unit files listed.

3.2.2. Displaying service status

To display detailed information about a service unit that corresponds to a system service, type the following at a shell prompt:

```
systemctl status name.service
```

Replace name with the name of the service unit you want to inspect (for example, gdm). This command displays the name of the selected service unit followed by its short description, one or more fields described in Table 3.5, “Available service unit information”, and if it is executed by the root user, also the most recent log entries.

Table 3.5. Available service unit information

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Loaded</td>
<td>Information whether the service unit has been loaded, the absolute path to the unit file, and a note whether the unit is enabled.</td>
</tr>
<tr>
<td>Active</td>
<td>Information whether the service unit is running followed by a time stamp.</td>
</tr>
<tr>
<td>Main PID</td>
<td>The PID of the corresponding system service followed by its name.</td>
</tr>
</tbody>
</table>
To only verify that a particular service unit is running, run the following command:

```
$ systemctl is-active name.service
```

Similarly, to determine whether a particular service unit is enabled, type:

```
$ systemctl is-enabled name.service
```

Note that both `systemctl is-active` and `systemctl is-enabled` return an exit status of 0 if the specified service unit is running or enabled. For information on how to list all currently loaded service units, see Listing services.

**Example 3.2. Displaying service status**

The service unit for the GNOME Display Manager is named `gdm.service`. To determine the current status of this service unit, type the following at a shell prompt:

```
[~]# systemctl status gdm.service
```

```
gdm.service - GNOME Display Manager
   Loaded: loaded (/usr/lib/systemd/system/gdm.service; enabled)
   Active: active (running) since Thu 2013-10-17 17:31:23 CEST; 5min ago
   Main PID: 1029 (gdm)
   CGroup: /system.slice/gdm.service
      └─ 1029 /usr/sbin/gdm
```

```
Oct 17 17:31:23 localhost systemd[1]: Started GNOME Display Manager.
```

**Example 3.3. Displaying services ordered to start before a service**

To determine what services are ordered to start before the specified service, type the following at a shell prompt:

```
[~]# systemctl list-dependencies --after gdm.service
```

```
gdm.service
   └─dbus.socket
   └─getty@tty1.service
   └─livesys.service
   └─plymouth-quit.service
```
Example 3.4. Displaying services ordered to start after a service

To determine what services are ordered to start after the specified service, type the following at a shell prompt:

```
~# systemctl list-dependencies --before gdm.service
```

gdm.service
  └─ dracut-shutdown.service
    └─ graphical.target
        └─ systemd-readahead-done.service
            └─ systemd-readahead-done.timer
                └─ systemd-update-utmp-runlevel.service
        └─ shutdown.target
            └─ systemd-reboot.service
                └─ final.target
                    └─ systemd-reboot.service

3.2.3. Starting a service

To start a service unit that corresponds to a system service, type the following at a shell prompt as `root`:

```
systemctl start name.service
```

Replace `name` with the name of the service unit you want to start (for example, `gdm`). This command starts the selected service unit in the current session. For information on how to enable a service unit to be started at boot time, see Enabling a service. For information on how to determine the status of a certain service unit, see Displaying service status.

Example 3.5. Starting a service

The service unit for the Apache HTTP Server is named `httpd.service`. To activate this service unit and start the `httpd` daemon in the current session, run the following command as `root`:

```
~# systemctl start httpd.service
```

3.2.4. Stopping a service

To stop a service unit that corresponds to a system service, type the following at a shell prompt as `root`:

```
systemctl stop name.service
```

Replace `name` with the name of the service unit you want to stop (for example, `bluetooth`). This command stops the selected service unit in the current session. For information on how to disable a
service unit and prevent it from being started at boot time, see Disabling a service. For information on how to determine the status of a certain service unit, see Displaying service status.

Example 3.6. Stopping a service

The service unit for the `bluetoothd` daemon is named `bluetooth.service`. To deactivate this service unit and stop the `bluetoothd` daemon in the current session, run the following command as `root`:

```
~]# systemctl stop bluetooth.service
```

3.2.5. Restarting a service

To restart a service unit that corresponds to a system service, type the following at a shell prompt as `root`:

```
systemctl restart name.service
```

Replace `name` with the name of the service unit you want to restart (for example, `httpd`). This command stops the selected service unit in the current session and immediately starts it again. Importantly, if the selected service unit is not running, this command starts it too. To tell `systemd` to restart a service unit only if the corresponding service is already running, run the following command as `root`:

```
systemctl try-restart name.service
```

Certain system services also allow you to reload their configuration without interrupting their execution. To do so, type as `root`:

```
systemctl reload name.service
```

Note that system services that do not support this feature ignore this command altogether. For convenience, the `systemctl` command also supports the `reload-or-restart` and `reload-or-try-restart` commands that restart such services instead. For information on how to determine the status of a certain service unit, see Displaying service status.

Example 3.7. Restarting a service

In order to prevent users from encountering unnecessary error messages or partially rendered web pages, the Apache HTTP Server allows you to edit and reload its configuration without the need to restart it and interrupt actively processed requests. To do so, type the following at a shell prompt as `root`:

```
~]# systemctl reload httpd.service
```

3.2.6. Enabling a service

To configure a service unit that corresponds to a system service to be automatically started at boot time, type the following at a shell prompt as `root`:

```
systemctl enable name.service
```
Replace name with the name of the service unit you want to enable (for example, **httpd**). This command reads the [Install] section of the selected service unit and creates appropriate symbolic links to the /usr/lib/systemd/system/name.service file in the /etc/systemd/system/ directory and its subdirectories. This command does not, however, rewrite links that already exist. If you want to ensure that the symbolic links are re-created, use the following command as root:

```bash
systemctl reenable name.service
```

This command enables the selected service unit and immediately enables it again. For information on how to determine whether a certain service unit is enabled to start at boot time, see Displaying service status. For information on how to start a service in the current session, see Starting a service.

### Example 3.8. Enabling a service

To configure the Apache HTTP Server to start automatically at boot time, run the following command as root:

```bash
~$ systemctl enable httpd.service
Created symlink from /etc/systemd/system/multi-user.target.wants/httpd.service to /usr/lib/systemd/system/httpd.service.
```

3.2.7. Disabling a service

To prevent a service unit that corresponds to a system service from being automatically started at boot time, type the following at a shell prompt as root:

```bash
systemctl disable name.service
```

This command replaces the /etc/systemd/system/name.service file with a symbolic link to /dev/null, rendering the actual unit file inaccessible to systemd. To revert this action and unmask a service unit, type as root:

```bash
systemctl unmask name.service
```

For information on how to determine whether a certain service unit is enabled to start at boot time, see Displaying service status. For information on how to stop a service in the current session, see Stopping a service.

### Example 3.9. Disabling a service

Example 3.6, “Stopping a service” illustrates how to stop the **bluetooth.service** unit in the current session. To prevent this service unit from starting at boot time, type the following at a shell prompt as root:

```bash
systemctl mask bluetooth.service
```

Red Hat Enterprise Linux 8 Configuring basic system settings
3.2.8. Starting a conflicting service

In systemd, positive and negative dependencies between services exist. Starting particular service may require starting one or more other services (positive dependency) or stopping one or more services (negative dependency).

When you attempt to start a new service, systemd resolves all dependencies automatically. Note that this is done without explicit notification to the user. If you are already running a service, and you attempt to start another service with a negative dependency, the first service is automatically stopped.

For example, if you are running the postfix service, and you try to start the sendmail service, systemd first automatically stops postfix, because these two services are conflicting and cannot run on the same port.

3.3. WORKING WITH SYSTEMD TARGETS

Previous versions of Red Hat Enterprise Linux, which were distributed with SysV init or Upstart, implemented a predefined set of runlevels that represented specific modes of operation. These runlevels were numbered from 0 to 6 and were defined by a selection of system services to be run when a particular runlevel was enabled by the system administrator. Starting with Red Hat Enterprise Linux 7, the concept of runlevels has been replaced with systemd targets.

Systemd targets are represented by target units. Target units end with the .target file extension and their only purpose is to group together other systemd units through a chain of dependencies. For example, the graphical.target unit, which is used to start a graphical session, starts system services such as the GNOME Display Manager (gdm.service) or Accounts Service (accounts-daemon.service) and also activates the multi-user.target unit. Similarly, the multi-user.target unit starts other essential system services such as NetworkManager (NetworkManager.service) or D-Bus (dbus.service) and activates another target unit named basic.target.

Red Hat Enterprise Linux 7 was distributed with a number of predefined targets that are more or less similar to the standard set of runlevels from the previous releases of this system. For compatibility reasons, it also provides aliases for these targets that directly map them to SysV runlevels. Table 3.6, “Comparison of SysV runlevels with systemd targets” provides a complete list of SysV runlevels and their corresponding systemd targets.

Table 3.6. Comparison of SysV runlevels with systemd targets

<table>
<thead>
<tr>
<th>Runlevel</th>
<th>Target Units</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>runlevel0.target, poweroff.target</td>
<td>Shut down and power off the system.</td>
</tr>
<tr>
<td>1</td>
<td>runlevel1.target, rescue.target</td>
<td>Set up a rescue shell.</td>
</tr>
<tr>
<td>2</td>
<td>runlevel2.target, multi-user.target</td>
<td>Set up a non-graphical multi-user system.</td>
</tr>
</tbody>
</table>
### Table 3.7. Comparison of SysV init commands with systemctl

<table>
<thead>
<tr>
<th>Old Command</th>
<th>New Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>runlevel</td>
<td>systemctl list-units --type</td>
<td>Lists currently loaded target units.</td>
</tr>
<tr>
<td></td>
<td>target</td>
<td></td>
</tr>
<tr>
<td>telinit runlevel</td>
<td>systemctl isolate</td>
<td>Changes the current target.</td>
</tr>
<tr>
<td></td>
<td>name.target</td>
<td></td>
</tr>
</tbody>
</table>

#### 3.3.1. Viewing the default target

To determine which target unit is used by default, run the following command:

```
systemctl get-default
```

This command resolves the symbolic link located at `/etc/systemd/system/default.target` and displays the result.

**Example 3.10. Viewing the default target**

To display the default target unit, type:

```
$ systemctl get-default
graphical.target
```

#### 3.3.2. Viewing the current target

To list all currently loaded target units, type the following command at a shell prompt:

```
systemctl list-units --type target
```
systemctl list-units --type target

For each target unit, this command displays its full name (UNIT) followed by a note whether the unit has been loaded (LOAD), its high-level (ACTIVE) and low-level (SUB) unit activation state, and a short description (DESCRIPTION).

By default, the systemctl list-units command displays only active units. If you want to list all loaded units regardless of their state, run this command with the --all or -a command line option:

systemctl list-units --type target --all

Example 3.11. Viewing the current target

To list all currently loaded target units, run:

```
~]$ systemctl list-units --type target
UNIT                  LOAD   ACTIVE SUB    DESCRIPTION
basic.target          loaded active active Basic System
cryptsetup.target     loaded active active Encrypted Volumes
getty.target          loaded active active Login Prompts
 graphical.target      loaded active active Graphical Interface
local-fs-pre.target   loaded active active Local File Systems (Pre)
local-fs.target       loaded active active Local File Systems
multi-user.target     loaded active active Multi-User System
network.target        loaded active active Network
paths.target          loaded active active Paths
remote-fs.target      loaded active active Remote File Systems
sockets.target        loaded active active Sockets
sound.target          loaded active active Sound Card
spice-vdagentd.target loaded active active Agent daemon for Spice guests
swap.target           loaded active active Swap
sysinit.target        loaded active active System Initialization
time-sync.target      loaded active active System Time Synchronized
timers.target         loaded active active Timers
```

LOAD = Reflects whether the unit definition was properly loaded.
ACTIVE = The high-level unit activation state, i.e. generalization of SUB.
SUB = The low-level unit activation state, values depend on unit type.

17 loaded units listed. Pass --all to see loaded but inactive units, too.
To show all installed unit files use 'systemctl list-unit-files'.

3.3.3. Changing the default target

To configure the system to use a different target unit by default, type the following at a shell prompt as root:

```
systemctl set-default name.target
```

Replace name with the name of the target unit you want to use by default (for example, multi-user). This command replaces the /etc/systemd/system/default.target file with a symbolic link to /usr/lib/systemd/system/name.target, where name is the name of the target unit you want to use.

---
Example 3.12. Changing the default target

To configure the system to use the `multi-user.target` unit by default, run the following command as `root`:

```bash
~]# systemctl set-default multi-user.target
rm '/etc/systemd/system/default.target'
ln -s '/usr/lib/systemd/system/multi-user.target' '/etc/systemd/system/default.target'
```

3.3.4. Changing the current target

To change to a different target unit in the current session, type the following at a shell prompt as `root`:

```
systemctl isolate name.target
```

Replace `name` with the name of the target unit you want to use (for example, `multi-user`). This command starts the target unit named `name` and all dependent units, and immediately stops all others.

Example 3.13. Changing the current target

To turn off the graphical user interface and change to the `multi-user.target` unit in the current session, run the following command as `root`:

```bash
~]# systemctl isolate multi-user.target
```

3.3.5. Changing to rescue mode

`Rescue mode` provides a convenient single-user environment and allows you to repair your system in situations when it is unable to complete a regular booting process. In rescue mode, the system attempts to mount all local file systems and start some important system services, but it does not activate network interfaces or allow more users to be logged into the system at the same time. Rescue mode requires the root password.

To change the current target and enter rescue mode in the current session, type the following at a shell prompt as `root`:

```
systemctl rescue
```

This command is similar to `systemctl isolate rescue.target`, but it also sends an informative message to all users that are currently logged into the system. To prevent `systemd` from sending this message, run this command with the `--no-wall` command line option:

```
systemctl --no-wall rescue
```

For information on how to enter emergency mode, see Section 3.3.6, “Changing to emergency mode”.

Example 3.14. Changing to rescue mode

To enter rescue mode in the current session, run the following command as `root`:
3.3.6. Changing to emergency mode

*Emergency mode* provides the most minimal environment possible and allows you to repair your system even in situations when the system is unable to enter rescue mode. In emergency mode, the system mounts the root file system only for reading, does not attempt to mount any other local file systems, does not activate network interfaces, and only starts a few essential services. Emergency mode requires the root password.

To change the current target and enter emergency mode, type the following at a shell prompt as **root**:

```
# systemctl emergency
```

This command is similar to **systemctl isolate emergency.target**, but it also sends an informative message to all users that are currently logged into the system. To prevent **systemd** from sending this message, run this command with the **--no-wall** command line option:

```
# systemctl --no-wall emergency
```

For information on how to enter rescue mode, see Section 3.3.5, “Changing to rescue mode”.

**Example 3.15. Changing to emergency mode**

To enter emergency mode without sending a message to all users that are currently logged into the system, run the following command as **root**:

```
~# systemctl --no-wall emergency
```

### 3.4. SHUTTING DOWN, SUSPENDING, AND HIBERNATING THE SYSTEM

In Red Hat Enterprise Linux 7, the **systemctl** utility replaced a number of power management commands used in previous versions of Red Hat Enterprise Linux. The commands listed in Table 3.8, “Comparison of power management commands with systemctl” are still available in the system for compatibility reasons, but it is advised that you use **systemctl** when possible.

**Table 3.8. Comparison of power management commands with systemctl**

<table>
<thead>
<tr>
<th>Old Command</th>
<th>New Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>halt</td>
<td>systemctl halt</td>
<td>Halts the system.</td>
</tr>
<tr>
<td>poweroff</td>
<td>systemctl poweroff</td>
<td>Powers off the system.</td>
</tr>
</tbody>
</table>
### 3.4.1. Shutting down the system

The `systemctl` utility provides commands for shutting down the system, however the traditional `shutdown` command is also supported. Although the `shutdown` command will call the `systemctl` utility to perform the shutdown, it has an advantage in that it also supports a time argument. This is particularly useful for scheduled maintenance and to allow more time for users to react to the warning that a system shutdown has been scheduled. The option to cancel the shutdown can also be an advantage.

#### Using systemctl commands

To shut down the system and power off the machine, type the following at a shell prompt as **root**:

```
systemctl poweroff
```

To shut down and halt the system without powering off the machine, run the following command as **root**:

```
systemctl halt
```

By default, running either of these commands causes `systemd` to send an informative message to all users that are currently logged into the system. To prevent `systemd` from sending this message, run the selected command with the `--no-wall` command line option, for example:

```
systemctl --no-wall poweroff
```

#### Using the shutdown command

To shut down the system and power off the machine at a certain time, use a command in the following format as **root**:

```
shutdown --poweroff hh:mm
```

Where `hh:mm` is the time in 24 hour clock format. The `/run/nologin` file is created 5 minutes before system shutdown to prevent new logins. When a time argument is used, an optional message, the `wall` message, can be appended to the command.

To shut down and halt the system after a delay, without powering off the machine, use a command in the following format as **root**:

```
shutdown --halt +m
```
Where \( +m \) is the delay time in minutes. The `now` keyword is an alias for `+0`.

A pending shutdown can be canceled by the `root` user as follows:

```
shutdown -c
```

See the `shutdown(8)` manual page for further command options.

### 3.4.2. Restarting the system

To restart the system, run the following command as `root`:

```
systemctl reboot
```

By default, this command causes `systemd` to send an informative message to all users that are currently logged into the system. To prevent `systemd` from sending this message, run this command with the `--no-wall` command line option:

```
systemctl --no-wall reboot
```

### 3.4.3. Suspending the system

To suspend the system, type the following at a shell prompt as `root`:

```
systemctl suspend
```

This command saves the system state in RAM and with the exception of the RAM module, powers off most of the devices in the machine. When you turn the machine back on, the system then restores its state from RAM without having to boot again. Because the system state is saved in RAM and not on the hard disk, restoring the system from suspend mode is significantly faster than restoring it from hibernation, but as a consequence, a suspended system state is also vulnerable to power outages.

For information on how to hibernate the system, see Section 3.4.4, “Hibernating the system”.

### 3.4.4. Hibernating the system

To hibernate the system, type the following at a shell prompt as `root`:

```
systemctl hibernate
```

This command saves the system state on the hard disk drive and powers off the machine. When you turn the machine back on, the system then restores its state from the saved data without having to boot again. Because the system state is saved on the hard disk and not in RAM, the machine does not have to maintain electrical power to the RAM module, but as a consequence, restoring the system from hibernation is significantly slower than restoring it from suspend mode.

To hibernate and suspend the system, run the following command as `root`:

```
systemctl hybrid-sleep
```

For information on how to suspend the system, see Section 3.4.3, “Suspending the system”.
3.5. WORKING WITH SYSTEMD UNIT FILES

A unit file contains configuration directives that describe the unit and define its behavior. Several systemctl commands work with unit files in the background. To make finer adjustments, system administrator must edit or create unit files manually. Table 3.1, “Systemd unit files locations” lists three main directories where unit files are stored on the system, the /etc/systemd/system/ directory is reserved for unit files created or customized by the system administrator.

Unit file names take the following form:

```
unit_name.type_extension
```

Here, `unit_name` stands for the name of the unit and `type_extension` identifies the unit type, see Table 3.2, “Available systemd unit types” for a complete list of unit types. For example, there usually is sshd.service as well as sshd.socket unit present on your system.

Unit files can be supplemented with a directory for additional configuration files. For example, to add custom configuration options to sshd.service, create the sshd.service.d/custom.conf file and insert additional directives there. For more information on configuration directories, see Section 3.5.4, “Modifying existing unit files”.

Also, the sshd.service.wants/ and sshd.service.requires/ directories can be created. These directories contain symbolic links to unit files that are dependencies of the sshd service. The symbolic links are automatically created either during installation according to [Install] unit file options or at runtime based on [Unit] options. It is also possible to create these directories and symbolic links manually. For more details on [Install] and [Unit] options, see the tables below.

Many unit file options can be set using the so called unit specifiers – wildcard strings that are dynamically replaced with unit parameters when the unit file is loaded. This enables creation of generic unit files that serve as templates for generating instantiated units. See Section 3.5.5, “Working with instantiated units” for details.

3.5.1. Understanding the unit file structure

Unit files typically consist of three sections:

- The [Unit] section – contains generic options that are not dependent on the type of the unit. These options provide unit description, specify the unit’s behavior, and set dependencies to other units. For a list of most frequently used [Unit] options, see Table 3.9, “Important [Unit] section options”.

- The [Unit type] section – if a unit has type-specific directives, these are grouped under a section named after the unit type. For example, service unit files contain the [Service] section.

- The [Install] section – contains information about unit installation used by systemctl enable and disable commands. For a list of options for the [Install] section, see Table 3.11, “Important [Install] section options”.

Table 3.9. Important [Unit] section options

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>[a] section, see the systemd.unit(5) manual page.</td>
<td></td>
</tr>
</tbody>
</table>

Red Hat Enterprise Linux 8 Configuring basic system settings
<table>
<thead>
<tr>
<th>Option[a] section, see the systemd.unit(5) manual page.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Description</strong></td>
<td>A meaningful description of the unit. This text is displayed for example in the output of the systemctl status command.</td>
</tr>
<tr>
<td><strong>Documentation</strong></td>
<td>Provides a list of URIs referencing documentation for the unit.</td>
</tr>
<tr>
<td><strong>After</strong>[b]</td>
<td>Defines the order in which units are started. The unit starts only after the units specified in After are active. Unlike Requires, After does not explicitly activate the specified units. The Before option has the opposite functionality to After.</td>
</tr>
<tr>
<td><strong>Requires</strong></td>
<td>Configures dependencies on other units. The units listed in Requires are activated together with the unit. If any of the required units fail to start, the unit is not activated.</td>
</tr>
<tr>
<td><strong>Wants</strong></td>
<td>Configures weaker dependencies than Requires. If any of the listed units does not start successfully, it has no impact on the unit activation. This is the recommended way to establish custom unit dependencies.</td>
</tr>
<tr>
<td><strong>Conflicts</strong></td>
<td>Configures negative dependencies, an opposite to Requires.</td>
</tr>
</tbody>
</table>

[a] For a complete list of options configurable in the [Unit]

[b] In most cases, it is sufficient to set only the ordering dependencies with After and Before unit file options. If you also set a requirement dependency with Wants (recommended) or Requires, the ordering dependency still needs to be specified. That is because ordering and requirement dependencies work independently from each other.

### Table 3.10. Important [Service] section options

<table>
<thead>
<tr>
<th>Option[a] section, see the systemd.service(5) manual page.</th>
<th>Description</th>
</tr>
</thead>
</table>

[a] For a complete list of options configurable in the [Service] section, see the systemd.service(5) manual page.
<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Type</strong></td>
<td>Configures the unit process startup type that affects the functionality of <strong>ExecStart</strong> and related options. One of: \n</td>
</tr>
<tr>
<td><strong>ExecStart</strong></td>
<td>Specifies commands or scripts to be executed when the unit is started. <strong>ExecStartPre</strong> and <strong>ExecStartPost</strong> specify custom commands to be executed before and after <strong>ExecStart</strong>. <strong>Type=oneshot</strong> enables specifying multiple custom commands that are then executed sequentially.</td>
</tr>
<tr>
<td><strong>ExecStop</strong></td>
<td>Specifies commands or scripts to be executed when the unit is stopped.</td>
</tr>
<tr>
<td><strong>ExecReload</strong></td>
<td>Specifies commands or scripts to be executed when the unit is reloaded.</td>
</tr>
<tr>
<td><strong>Restart</strong></td>
<td>With this option enabled, the service is restarted after its process exits, with the exception of a clean stop by the <strong>systemctl</strong> command.</td>
</tr>
<tr>
<td><strong>RemainAfterExit</strong></td>
<td>If set to True, the service is considered active even when all its processes exited. Default value is False. This option is especially useful if <strong>Type=oneshot</strong> is configured.</td>
</tr>
</tbody>
</table>
A whole range of options that can be used to fine tune the unit configuration. The below example shows a service unit installed on the system. Moreover, unit file options can be defined in a way that enables dynamic creation of units as described in Working with instantiated units.

**Example 3.16. postfix.service unit file**

What follows is the content of the `/usr/lib/systemd/system/postfix.service` unit file as currently provided by the postfix package:

```
[Unit]
Description=Postfix Mail Transport Agent
After=syslog.target network.target
Conflicts=sendmail.service exim.service

[Service]
Type=forking
PIDFile=/var/spool/postfix/pid/master.pid
```
3.5.2. Creating custom unit files

There are several use cases for creating unit files from scratch: you could run a custom daemon, create a second instance of some existing service (as in Creating a second instance of the sshd service), or import a SysV init script (more in Converting SysV init scripts to unit files). On the other hand, if you intend just to modify or extend the behavior of an existing unit, use the instructions from Modifying existing unit files. The following procedure describes the general process of creating a custom service:

1. Prepare the executable file with the custom service. This can be a custom-created script, or an executable delivered by a software provider. If required, prepare a PID file to hold a constant PID for the main process of the custom service. It is also possible to include environment files to store shell variables for the service. Make sure the source script is executable (by executing the `chmod a+x`) and is not interactive.

2. Create a unit file in the `/etc/systemd/system/` directory and make sure it has correct file permissions. Execute as `root`:

   ```bash
   touch /etc/systemd/system/name.service
   chmod 664 /etc/systemd/system/name.service
   ```

   Replace `name` with a name of the service to be created. Note that file does not need to be executable.

3. Open the `name.service` file created in the previous step, and add the service configuration options. There is a variety of options that can be used depending on the type of service you wish to create, see Section 3.5.1, "Understanding the unit file structure". The following is an example unit configuration for a network-related service:

   ```ini
   [Unit]
   Description=service_description
   After=network.target

   [Service]
   ExecStart=path_to_executable
   Type=forking
   PIDFile=path_to_pidfile
   ```
[Install]
WantedBy=default.target

Where:

- `service_description` is an informative description that is displayed in journal log files and in the output of the `systemctl status` command.
- The `After` setting ensures that the service is started only after the network is running. Add a space-separated list of other relevant services or targets.
- `path_to_executable` stands for the path to the actual service executable.
- `Type=forking` is used for daemons that make the fork system call. The main process of the service is created with the PID specified in `path_to_pidfile`. Find other startup types in Table 3.10, “Important [Service] section options”.
- `WantedBy` states the target or targets that the service should be started under. Think of these targets as a replacement of the older concept of runlevels, see Section 3.3, “Working with systemd targets” for details.

4. Notify `systemd` that a new `name.service` file exists by executing the following command as `root`:

```bash
systemctl daemon-reload
systemctl start name.service
```

**WARNING**
Always run the `systemctl daemon-reload` command after creating new unit files or modifying existing unit files. Otherwise, the `systemctl start` or `systemctl enable` commands could fail due to a mismatch between states of `systemd` and actual service unit files on disk. Note, that on systems with a large number of units this can take a long time, as the state of each unit has to be serialized and subsequently deserialized during the reload.

**Example 3.17. Creating the emacs.service file**

When using the Emacs text editor, it is often faster and more convenient to have it running in the background instead of starting a new instance of the program whenever editing a file. The following steps show how to create a unit file for Emacs, so that it can be handled like a service.

1. Create a unit file in the `/etc/systemd/system/` directory and make sure it has the correct file permissions. Execute as `root`:

   ```bash
   ~]# touch /etc/systemd/system/emacs.service
   ~]# chmod 664 /etc/systemd/system/emacs.service
   ```
2. Add the following content to the file:

```
[Unit]
Description=Emacs: the extensible, self-documenting text editor

[Service]
Type=forking
ExecStart=/usr/bin/emacs --daemon
ExecStop=/usr/bin/emacsclient --eval "(kill-emacs)"
Environment=SSH_AUTH_SOCK=%t/keyring/ssh
Restart=always

[Install]
WantedBy=default.target
```

With the above configuration, the `/usr/bin/emacs` executable is started in daemon mode on service start. The `SSH_AUTH_SOCK` environment variable is set using the "%t" unit specifier that stands for the runtime directory. The service also restarts the emacs process if it exits unexpectedly.

3. Execute the following commands to reload the configuration and start the custom service:

```
~]# systemctl daemon-reload
~]# systemctl start emacs.service
```

As the editor is now registered as a systemd service, you can use all standard `systemctl` commands. For example, run `systemctl status emacs` to display the editor’s status or `systemctl enable emacs` to make the editor start automatically on system boot.

**Example 3.18. Creating a second instance of the sshd service**

System Administrators often need to configure and run multiple instances of a service. This is done by creating copies of the original service configuration files and modifying certain parameters to avoid conflicts with the primary instance of the service. The following procedure shows how to create a second instance of the `sshd` service:

1. Create a copy of the `sshd_config` file that will be used by the second daemon:

```
~]# cp /etc/ssh/sshd{-second}_config
```

2. Edit the `sshd-second_config` file created in the previous step to assign a different port number and PID file to the second daemon:

```
Port 22220
PidFile /var/run/sshd-second.pid
```

See the `sshd_config`(5) manual page for more information on `Port` and `PidFile` options. Make sure the port you choose is not in use by any other service. The PID file does not have to exist before running the service, it is generated automatically on service start.

3. Create a copy of the systemd unit file for the `sshd` service:
4. Alter the `sshd-second.service` created in the previous step as follows:
   a. Modify the `Description` option:
      ```
      Description=OpenSSH server second instance daemon
      ```
   b. Add `sshd.service` to services specified in the `After` option, so that the second instance starts only after the first one has already started:
      ```
      After=syslog.target network.target auditd.service sshd.service
      ```
   c. The first instance of `sshd` includes key generation, therefore remove the `ExecStartPre=/usr/sbin/sshd-keygen` line.
   d. Add the `-f /etc/ssh/sshd-second_config` parameter to the `sshd` command, so that the alternative configuration file is used:
      ```
      ExecStart=/usr/sbin/sshd -D -f /etc/ssh/sshd-second_config $OPTIONS
      ```
   e. After the above modifications, the `sshd-second.service` should look as follows:
      ```
      [Unit]
      Description=OpenSSH server second instance daemon
      After=syslog.target network.target auditd.service sshd.service

      [Service]
      EnvironmentFile=/etc/sysconfig/sshd
      ExecStart=/usr/sbin/sshd -D -f /etc/ssh/sshd-second_config $OPTIONS
      ExecReload=/bin/kill -HUP $MAINPID
      KillMode=process
      Restart=on-failure
      RestartSec=42s

      [Install]
      WantedBy=multi-user.target
      ```

5. If using SELinux, add the port for the second instance of `sshd` to SSH ports, otherwise the second instance of `sshd` will be rejected to bind to the port:
   ```
   ~]$ semanage port -a -t ssh_port_t -p tcp 22220
   ```

6. Enable `sshd-second.service`, so that it starts automatically upon boot:
   ```
   ~]$ systemctl enable sshd-second.service
   ```
   Verify if the `sshd-second.service` is running by using the `systemctl status` command. Also, verify if the port is enabled correctly by connecting to the service:
   ```
   ~]$ ssh -p 22220 user@server
   ```
If the firewall is in use, make sure that it is configured appropriately in order to allow connections to the second instance of sshd.

To learn how to properly choose a target for ordering and dependencies of your custom unit files, see the following articles

- How to write a service unit file which enforces that particular services have to be started
- How to decide what dependencies a systemd service unit definition should have

Additional information with some real-world examples of cases triggered by the ordering and dependencies in a unit file is available in Red Hat Knowledgebase article Is there any useful information about writing unit files?

If you want to set limits for services started by systemd, see the Red Hat Knowledgebase article How to set limits for services in RHEL 7 and systemd. These limits need to be set in the service’s unit file. Note that systemd ignores limits set in the /etc/security/limits.conf and /etc/security/limits.d/*.conf configuration files. The limits defined in these files are set by PAM when starting a login session, but daemons started by systemd do not use PAM login sessions.

3.5.3. Converting SysV init scripts to unit files

Before taking time to convert a SysV init script to a unit file, make sure that the conversion was not already done elsewhere. All core services installed on Red Hat Enterprise Linux come with default unit files, and the same applies for many third-party software packages.

Converting an init script to a unit file requires analyzing the script and extracting the necessary information from it. Based on this data you can create a unit file. As init scripts can vary greatly depending on the type of the service, you might need to employ more configuration options for translation than outlined in this chapter. Note that some levels of customization that were available with init scripts are no longer supported by systemd units.

The majority of information needed for conversion is provided in the script’s header. The following example shows the opening section of the init script used to start the postfix service on Red Hat Enterprise Linux 6:

```
/bin/bash # postfix Postfix Mail Transfer Agent # chkconfig: 2345 80 30 # description: Postfix is a Mail Transport Agent, which is the program that moves mail from one machine to another. # processname: master # pidfile: /var/spool/postfix/pid/master.pid # config: /etc/postfix/main.cf # config: /etc/postfix/master.cf BEGIN INIT INFO # Provides: postfix MTA # Required-Start: $local_fs $network $remote_fs # Required-Stop: $local_fs $network $remote_fs # Default-Start: 2 3 4 5 # Default-Stop: 0 1 6 # Short-Description: start and stop postfix # Description: Postfix is a Mail Transport Agent, which is the program that moves mail from one machine to another. # END INIT INFO
```

In the above example, only lines starting with # chkconfig and # description are mandatory, so you might not find the rest in different init files. The text enclosed between the BEGIN INIT INFO and END INIT INFO lines is called Linux Standard Base (LSB) header. If specified, LSB headers contain directives defining the service description, dependencies, and default runlevels. What follows is an overview of analytic tasks aiming to collect the data needed for a new unit file. The postfix init script is used as an example, see the resulting postfix unit file in Example 3.16, “postfix.service unit file”.

Finding the service description
Find descriptive information about the script on the line starting with `#description`. Use this description together with the service name in the **Description** option in the [Unit] section of the unit file. The LSB header might contain similar data on the **#Short-Description** and **#Description** lines.

### Finding service dependencies

The LSB header might contain several directives that form dependencies between services. Most of them are translatable to systemd unit options, see Table 3.12, “Dependency options from the LSB header”

<table>
<thead>
<tr>
<th>LSB Option</th>
<th>Description</th>
<th>Unit File Equivalent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Provides</td>
<td>Specifies the boot facility name of the service, that can be referenced in other init scripts (with the “$” prefix). This is no longer needed as unit files refer to other units by their file names.</td>
<td>–</td>
</tr>
<tr>
<td>Required-Start</td>
<td>Contains boot facility names of required services. This is translated as an ordering dependency, boot facility names are replaced with unit file names of corresponding services or targets they belong to. For example, in case of <strong>postfix</strong>, the Required-Start dependency on <code>$network</code> was translated to the After dependency on <code>network.target</code>.</td>
<td><strong>After, Before</strong></td>
</tr>
<tr>
<td>Should-Start</td>
<td>Constitutes weaker dependencies than Required-Start. Failed Should-Start dependencies do not affect the service startup.</td>
<td><strong>After, Before</strong></td>
</tr>
<tr>
<td>Required-Stop, Should-Stop</td>
<td>Constitute negative dependencies.</td>
<td><strong>Conflicts</strong></td>
</tr>
</tbody>
</table>

### Finding default targets of the service

The line starting with `#chkconfig` contains three numerical values. The most important is the first number that represents the default runlevels in which the service is started. Map these runlevels to equivalent systemd targets. Then list these targets in the **WantedBy** option in the [Install] section of the unit file. For example, **postfix** was previously started in runlevels 2, 3, 4, and 5, which translates to multi-user.target and graphical.target. Note that the graphical.target depends on multiuser.target, therefore it is not necessary to specify both, as in Example 3.16, “postfix.service unit file”. You might find information on default and forbidden runlevels also at **#Default-Start** and **#Default-Stop** lines in the LSB header.

The other two values specified on the `#chkconfig` line represent startup and shutdown priorities of the init script. These values are interpreted by **systemd** if it loads the init script, but there is no unit file equivalent.
Finding files used by the service
Init scripts require loading a function library from a dedicated directory and allow importing configuration, environment, and PID files. Environment variables are specified on the line starting with `#config` in the init script header, which translates to the `EnvironmentFile` unit file option. The PID file specified on the `#pidfile` init script line is imported to the unit file with the `PIDFile` option.

The key information that is not included in the init script header is the path to the service executable, and potentially some other files required by the service. In previous versions of Red Hat Enterprise Linux, init scripts used a Bash case statement to define the behavior of the service on default actions, such as `start`, `stop`, or `restart`, as well as custom-defined actions. The following excerpt from the `postfix` init script shows the block of code to be executed at service start.

```bash
conf_check() {
    [ -x /usr/sbin/postfix ] || exit 5
    [ -d /etc/postfix ] || exit 6
    [ -d /var/spool/postfix ] || exit 5
}
make_aliasesdb() {
    if [ "$(/usr/sbin/postconf -h alias_database)" == "hash:/etc/aliases" ]
    then
        # /etc/aliases.db might be used by other MTA, make sure nothing
        # has touched it since our last newaliases call
        [ /etc/aliases -nt /etc/aliases.db ] ||
        [ "$ALIASESDB_STAMP" -nt /etc/aliases.db ] ||
        [ "$ALIASESDB_STAMP" -ot /etc/aliases.db ] || return
        /usr/bin/newaliases
        touch -r /etc/aliases.db "$ALIASESDB_STAMP"
    else
        /usr/bin/newaliases
    fi
}
start() {
    [ "$EUID" != "0" ] && exit 4
    # Check that networking is up.
    [ ${NETWORKING} = "no" ] && exit 1
    conf_check
    # Start daemons.
    echo -n "$Starting postfix: "
    make_aliasesdb >/dev/null 2>&1
    [ -x $CHROOT_UPDATE ] && $CHROOT_UPDATE
    /usr/sbin/postfix start >/dev/null 2>&1 && success || failure "$prog start"
    RETVAL=$?
    [ $RETVAL -eq 0 ] && touch $lockfile
    echo
    return $RETVAL
}
```

The extensibility of the init script allowed specifying two custom functions, `conf_check()` and `make_aliasesdb()`, that are called from the `start()` function block. On closer look, several external files and directories are mentioned in the above code: the main service executable `/usr/sbin/postfix`, the `/etc/postfix/` and `/var/spool/postfix/` configuration directories, as well as the `/usr/sbin/postconf/` directory.

**Systemd** supports only the predefined actions, but enables executing custom executables with
The `ExecStart`, `ExecStartPre`, `ExecStartPost`, `ExecStop`, and `ExecReload` options. The `/usr/sbin/postfix` together with supporting scripts are executed on service start. Consult the `postfix` unit file at Example 3.16, “postfix.service unit file”.

Converting complex init scripts requires understanding the purpose of every statement in the script. Some of the statements are specific to the operating system version, therefore you do not need to translate them. On the other hand, some adjustments might be needed in the new environment, both in unit file as well as in the service executable and supporting files.

### 3.5.4. Modifying existing unit files

Services installed on the system come with default unit files that are stored in the `/usr/lib/systemd/system/` directory. System Administrators should not modify these files directly, therefore any customization must be confined to configuration files in the `/etc/systemd/system/` directory. Depending on the extent of the required changes, pick one of the following approaches:

- Create a directory for supplementary configuration files at `/etc/systemd/system/unit.d/`. This method is recommended for most use cases. It enables extending the default configuration with additional functionality, while still referring to the original unit file. Changes to the default unit introduced with a package upgrade are therefore applied automatically. See Extending the default unit configuration for more information.

- Create a copy of the original unit file `/usr/lib/systemd/system/` in `/etc/systemd/system/` and make changes there. The copy overrides the original file, therefore changes introduced with the package update are not applied. This method is useful for making significant unit changes that should persist regardless of package updates. See Overriding the default unit configuration for details.

In order to return to the default configuration of the unit, just delete custom-created configuration files in `/etc/systemd/system/`. To apply changes to unit files without rebooting the system, execute:

```
systemctl daemon-reload
```

The `daemon-reload` option reloads all unit files and recreates the entire dependency tree, which is needed to immediately apply any change to a unit file. As an alternative, you can achieve the same result with the following command, which must be executed under the `root` user:

```
init q
```

Also, if the modified unit file belongs to a running service, this service must be restarted to accept new settings:

```
systemctl restart name.service
```
IMPORTANT

To modify properties, such as dependencies or timeouts, of a service that is handled by a SysV initscript, do not modify the initscript itself. Instead, create a systemd drop-in configuration file for the service as described in Extending the default unit configuration and Overriding the default unit configuration. Then manage this service in the same way as a normal systemd service.

For example, to extend the configuration of the network service, do not modify the /etc/rc.d/init.d/network initscript file. Instead, create new directory /etc/systemd/system/network.service.d/ and a systemd drop-in file /etc/systemd/system/network.service.d/my_config.conf. Then, put the modified values into the drop-in file. Note: systemd knows the network service as network.service, which is why the created directory must be called network.service.d

Extending the default unit configuration

To extend the default unit file with additional configuration options, first create a configuration directory in /etc/systemd/system/. If extending a service unit, execute the following command as root:

```
mkdir /etc/systemd/system/name.service.d/
```

Replace name with the name of the service you want to extend. The above syntax applies to all unit types.

Create a configuration file in the directory made in the previous step. Note that the file name must end with the .conf suffix. Type:

```
touch /etc/systemd/system/name.service.d/config_name.conf
```

Replace config_name with the name of the configuration file. This file adheres to the normal unit file structure, therefore all directives must be specified under appropriate sections, see Section 3.5.1, “Understanding the unit file structure”.

For example, to add a custom dependency, create a configuration file with the following content:

```
[Unit]
Requires=new_dependency
After=new_dependency
```

Where new_dependency stands for the unit to be marked as a dependency. Another example is a configuration file that restarts the service after its main process exited, with a delay of 30 seconds:

```
[Service]
Restart=always
RestartSec=30
```

It is recommended to create small configuration files focused only on one task. Such files can be easily moved or linked to configuration directories of other services.

To apply changes made to the unit, execute as root:

```
systemctl daemon-reload
systemctl restart name.service
```
Example 3.19. Extending the httpd.service configuration

To modify the httpd.service unit so that a custom shell script is automatically executed when starting the Apache service, perform the following steps. First, create a directory and a custom configuration file:

```bash
~]# mkdir /etc/systemd/system/httpd.service.d/
~]# touch /etc/systemd/system/httpd.service.d/custom_script.conf
```

Provided that the script you want to start automatically with Apache is located at `/usr/local/bin/custom.sh`, insert the following text to the `custom_script.conf` file:

```ini
[Service]
ExecStartPost=/usr/local/bin/custom.sh
```

To apply the unit changes, execute:

```bash
~]# systemctl daemon-reload
~]# systemctl restart httpd.service
```

**NOTE**

The configuration files from configuration directories in `/etc/systemd/system/` take precedence over unit files in `/usr/lib/systemd/system/`. Therefore, if the configuration files contain an option that can be specified only once, such as `Description` or `ExecStart`, the default value of this option is overridden. Note that in the output of the `systemd-delta` command, described in Monitoring overridden units, such units are always marked as `[EXTENDED]`, even though in sum, certain options are actually overridden.

**Overriding the default unit configuration**

To make changes that will persist after updating the package that provides the unit file, first copy the file to the `/etc/systemd/system/` directory. To do so, execute the following command as `root`:

```bash
cp /usr/lib/systemd/system/name.service /etc/systemd/system/name.service
```

Where `name` stands for the name of the service unit you wish to modify. The above syntax applies to all unit types.

Open the copied file with a text editor, and make the desired changes. To apply the unit changes, execute as `root`:

```bash
systemctl daemon-reload
systemctl restart name.service
```

Example 3.20. Changing the timeout limit
You can specify a timeout value per service to prevent a malfunctioning service from freezing the system. Otherwise, timeout is set by default to 90 seconds for normal services and to 300 seconds for SysV-compatible services.

For example, to extend timeout limit for the `httpd` service:

1. Copy the `httpd` unit file to the `/etc/systemd/system` directory:
   ```bash
   cp /usr/lib/systemd/system/httpd.service /etc/systemd/system/httpd.service
   ```

2. Open file `/etc/systemd/system/httpd.service` and specify the `TimeoutStartUSec` value in the `[Service]` section:
   ```bash
   ...[Service]...
   PrivateTmp=true
   TimeoutStartSec=10
   ...[Install]
   WantedBy=multi-user.target
   ...
   ```

3. Reload the `systemd` daemon:
   ```bash
   systemctl daemon-reload
   ```

4. Optional. Verify the new timeout value:
   ```bash
   systemctl show httpd -p TimeoutStartUSec
   ```

**NOTE**

To change the timeout limit globally, input the `DefaultTimeoutStartSec` in the `/etc/systemd/system.conf` file.

---

### Monitoring overriden units

To display an overview of overridden or modified unit files, use the following command:

```bash
systemd-delta
```

For example, the output of the above command can look as follows:

```bash
[EQUIVALENT] /etc/systemd/system/default.target → /usr/lib/systemd/system/default.target
[OVERRIDDEN] /etc/systemd/system/autofs.service → /usr/lib/systemd/system/autofs.service

+ /etc/systemd/system/autofs.service 2014-11-21 10:00:58.513568275 -0500
@@ -8,7 +8,8 @@
    ExecStart=/usr/sbin/automount $OPTIONS --pid-file /run/autofs.pid
    ExecReload=/usr/bin/kill -HUP $MAINPID
```
Table 3.13, “systemd-delta difference types” lists override types that can appear in the output of `systemd-delta`. Note that if a file is overridden, `systemd-delta` by default displays a summary of changes similar to the output of the `diff` command.

Table 3.13. systemd-delta difference types

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>[MASKED]</td>
<td>Masked unit files, see Section 3.2.7, “Disabling a service” for description of unit masking.</td>
</tr>
<tr>
<td>[EQUIVALENT]</td>
<td>Unmodified copies that override the original files but do not differ in content, typically symbolic links.</td>
</tr>
<tr>
<td>[REDIRECTED]</td>
<td>Files that are redirected to another file.</td>
</tr>
<tr>
<td>[OVERRIDEN]</td>
<td>Overridden and changed files.</td>
</tr>
<tr>
<td>[EXTENDED]</td>
<td>Files that are extended with .conf files in the <code>/etc/systemd/system/unit.d/</code> directory.</td>
</tr>
<tr>
<td>[UNCHANGED]</td>
<td>Unmodified files are displayed only when the --&gt; type=unchanged option is used.</td>
</tr>
</tbody>
</table>

It is good practice to run `systemd-delta` after system update to check if there are any updates to the default units that are currently overridden by custom configuration. It is also possible to limit the output only to a certain difference type. For example, to view just the overridden units, execute:

```
systemd-delta --type=overridden
```

If you want to edit a unit file and automatically create a drop-in file with the submitted changes, use the following command:

```
~]# systemctl edit unit_name.type_extension
```

To dump the unit configuration applying all overrides and drop-ins, use this command:

```
~]# systemctl cat unit_name.type_extension
```
Replace the `unit_name.type_extension` by the name of the required unit and its type, for example `tuned.service`.

### 3.5.5. Working with instantiated units

It is possible to instantiate multiple units from a single template configuration file at runtime. The `@` character is used to mark the template and to associate units with it. Instantiated units can be started from another unit file (using `Requires` or `Wants` options), or with the `systemctl start` command. Instantiated service units are named the following way:

```
template_name@instance_name.service
```

Where `template_name` stands for the name of the template configuration file. Replace `instance_name` with the name for the unit instance. Several instances can point to the same template file with configuration options common for all instances of the unit. Template unit name has the form of:

```
unit_name@.service
```

For example, the following `Wants` setting in a unit file:

```
Wants=getty@ttyA.service getty@ttyB.service
```

first makes systemd search for given service units. If no such units are found, the part between `@` and the type suffix is ignored and `systemd` searches for the `getty@.service` file, reads the configuration from it, and starts the services.

Wildcard characters, called `unit specifiers`, can be used in any unit configuration file. Unit specifiers substitute certain unit parameters and are interpreted at runtime. Table 3.14, “Important unit specifiers” lists unit specifiers that are particularly useful for template units.

#### Table 3.14. Important unit specifiers

<table>
<thead>
<tr>
<th>Unit Specifier</th>
<th>Meaning</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>%n</code></td>
<td>Full unit name</td>
<td>Stands for the full unit name including the type suffix. <code>%N</code> has the same meaning but also replaces the forbidden characters with ASCII codes.</td>
</tr>
<tr>
<td><code>%p</code></td>
<td>Prefix name</td>
<td>Stands for a unit name with type suffix removed. For instantiated units <code>%p</code> stands for the part of the unit name before the <code>@</code> character.</td>
</tr>
<tr>
<td><code>%i</code></td>
<td>Instance name</td>
<td>Is the part of the instantiated unit name between the <code>@</code> character and the type suffix. <code>%I</code> has the same meaning but also replaces the forbidden characters for ASCII codes.</td>
</tr>
<tr>
<td>Unit Specifier</td>
<td>Meaning</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>-----------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>%H</td>
<td>Host name</td>
<td>Stands for the hostname of the running system at the point in time the unit configuration is loaded.</td>
</tr>
<tr>
<td>%t</td>
<td>Runtime directory</td>
<td>Represents the runtime directory, which is either /run for the root user, or the value of the XDG_RUNTIME_DIR variable for unprivileged users.</td>
</tr>
</tbody>
</table>

For a complete list of unit specifiers, see the `systemd.unit(5)` manual page.

For example, the `getty@.service` template contains the following directives:

```ini
[Unit]
Description=Getty on %I
...

[Service]
ExecStart=/sbin/agetty --noclear %I $TERM
...
```

When the `getty@ttyA.service` and `getty@ttyB.service` are instantiated from the above template, `Description=` is resolved as `Getty on ttyA` and `Getty on ttyB`.

### 3.6. OPTIMIZING SYSTEMD TO SHORTEN THE BOOT TIME

There is a list of systemd unit files that are enabled by default. System services that are defined by these unit files are automatically run at boot, which influences the boot time.

This section describes:

- The tools to examine system boot performance.
- The purpose of systemd units enabled by default, and circumstances under which you can safely disable such systemd units in order to shorten the boot time.

#### 3.6.1. Examining system boot performance

To examine system boot performance, you can use the `systemd-analyze` command. This command has many options available. However, this section covers only the selected ones that may be important for systemd tuning in order to shorten the boot time.

For a complete list and detailed description of all options, see the `systemd-analyze` man page.

**Prerequisites**

Before starting to examine systemd in order to tune the boot time, you may want to list all enabled services:

```
~$ systemctl list-unit-files --state=enabled
```
Analyzing overall boot time
Procedure

- For the overall information about the time that the last successful boot took, use:

```
$ systemd-analyze
```

Analyzing unit initialization time
Procedure

- For the information about the initialization time of each systemd unit, use:

```
$ systemd-analyze blame
```

The output lists the units in descending order according to the time they took to initialize during the last successful boot.

Identifying critical units
Procedure

- To identify the units that took most time to initialize at the last successful boot, use:

```
$ systemd-analyze critical-chain
```

The output highlights the units that critically slow down the boot with the red color.

### Figure 3.1. The output of the `systemd-analyze critical-chain` command

```bash
[admin@localhost ~]$ systemd-analyze critical-chain

graphical.target @19.706s
└── multi-user.target @19.706s
    └── systemd-update-utmp.service @3.219s +311ms
        └── network.target @9.614s +759ms
            └── update-rc.d.service @16.025s +125ms
                └── auditd.service @2.446s
                    └── iscsi-iso.socket @2.446s
                        └── sysinit.target @2.446s
                            └── systemctl-start-stop.service @2.446s +100ms
```

#### 3.6.2. A guide to selecting services that can be safely disabled

If you find the boot time of your system long, you can shorten it by disabling some of the services enabled on boot by default.

To list such services, run:

```
$ systemctl list-unit-files --state=enabled
```

To disable a service, run:
However, certain services must stay enabled in order that your operating system is safe and functions in the way you need.

You can use the table below as a guide to selecting the services that you can safely disable. The table lists all services enabled by default on a minimal installation of Red Hat Enterprise Linux 8, and for each service it states whether this service can be safely disabled.

The table also provides more information about the circumstances under which the service can be disabled, or the reason why you should not disable the service.

Table 3.15. Services enabled by default on a minimal installation of RHEL 8

<table>
<thead>
<tr>
<th>Service name</th>
<th>Can it be disabled?</th>
<th>More information</th>
</tr>
</thead>
<tbody>
<tr>
<td>auditd.service</td>
<td>yes</td>
<td>Disable <code>auditd.service</code> only if you do not need audit messages from the kernel. Be aware that if you disable <code>auditd.service</code>, the <code>/var/log/audit/audit.log</code> file is not produced. Consequently, you are not able to retroactively review some commonly-reviewed actions or events, such as user logins, service starts or password changes. Also note that auditd has two parts: a kernel part, and a service itself. By using the <code>systemctl disable auditd</code> command, you only disable the service, but not the kernel part. To disable system auditing in its entirety, set <code>audit=0</code> on kernel command line.</td>
</tr>
<tr>
<td>autovt.service</td>
<td>no</td>
<td>This service runs only when it is really needed, so it does not need to be disabled.</td>
</tr>
<tr>
<td>crond.service</td>
<td>yes</td>
<td>Be aware that no items from crontab will run if you disable <code>crond.service</code>.</td>
</tr>
<tr>
<td>dbus-org.fedoraproject.FirewallD1.service</td>
<td>yes</td>
<td>A symlink to <code>firewalld.service</code></td>
</tr>
<tr>
<td>dbus-org.freedesktop.NetworkManager.service</td>
<td>yes</td>
<td>A symlink to <code>NetworkManager.service</code></td>
</tr>
<tr>
<td>dbus-org.freedesktop.nm-dispatcher.service</td>
<td>yes</td>
<td>A symlink to <code>NetworkManager-dispatcher.service</code></td>
</tr>
<tr>
<td>firewalld.service</td>
<td>yes</td>
<td>Disable <code>firewalld.service</code> only if you do not need firewall.</td>
</tr>
<tr>
<td>Service name</td>
<td>Can it be disabled?</td>
<td>More information</td>
</tr>
<tr>
<td>----------------------------</td>
<td>---------------------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>getty.service</td>
<td>no</td>
<td>This service runs only when it is really needed, so it does not need to be disabled.</td>
</tr>
<tr>
<td>import-state.service</td>
<td>yes</td>
<td>Disable import-state.service only if you do not need to boot from a network storage.</td>
</tr>
<tr>
<td>irqbalance.service</td>
<td>yes</td>
<td>Disable irqbalance.service only if you have just one CPU. Do not disable irqbalance.service on systems with multiple CPUs.</td>
</tr>
<tr>
<td>kdump.service</td>
<td>yes</td>
<td>Disable kdump.service only if you do not need reports from kernel crashes.</td>
</tr>
<tr>
<td>loadmodules.service</td>
<td>yes</td>
<td>This service is not started unless the /etc/rc.modules or /etc/sysconfig/modules directory exists, which means that it is not started on a minimal RHEL 8 installation.</td>
</tr>
<tr>
<td>lvm2-monitor.service</td>
<td>yes</td>
<td>Disable lvm2-monitor.service only if you do not use Logical Volume Manager (LVM).</td>
</tr>
<tr>
<td>microcode.service</td>
<td>no</td>
<td>Do not be disable the service because it provides updates of the microcode software in CPU.</td>
</tr>
<tr>
<td>NetworkManager-dispatcher.service</td>
<td>yes</td>
<td>Disable NetworkManager-dispatcher.service only if you do not need notifications on network configuration changes (for example in static networks).</td>
</tr>
<tr>
<td>NetworkManager-wait-online.service</td>
<td>yes</td>
<td>Disable NetworkManager-wait-online.service only if you do not need working network connection available right after the boot. If the service is enabled, the system does not finish the boot before the network connection is working. This may prolong the boot time significantly.</td>
</tr>
<tr>
<td>NetworkManager.service</td>
<td>yes</td>
<td>Disable NetworkManager.service only if you do not need connection to a network.</td>
</tr>
<tr>
<td>nis-domainname.service</td>
<td>yes</td>
<td>Disable nis-domainname.service only if you do not use Network Information Service (NIS).</td>
</tr>
<tr>
<td>rhsmcertd.service</td>
<td>no</td>
<td></td>
</tr>
<tr>
<td>Service name</td>
<td>Can it be disabled?</td>
<td>More information</td>
</tr>
<tr>
<td>----------------------------</td>
<td>---------------------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>rngd.service</td>
<td>yes</td>
<td>Disable <strong>rngd.service</strong> only if you do not need a lot of entropy on your system, or you do not have any sort of hardware generator. Note that the service is necessary in environments that require a lot of good entropy, such as systems used for generation of X.509 certificates (for example the FreeIPA server).</td>
</tr>
<tr>
<td>rsyslog.service</td>
<td>yes</td>
<td>Disable <strong>rsyslog.service</strong> only if you do not need persistent logs, or you set <strong>systemd-journald</strong> to persistent mode.</td>
</tr>
<tr>
<td>selinux-autorelabel-mark.service</td>
<td>yes</td>
<td>Disable <strong>selinux-autorelabel-mark.service</strong> only if you do not use SELinux.</td>
</tr>
<tr>
<td>sshd.service</td>
<td>yes</td>
<td>Disable <strong>sshd.service</strong> only if you do not need remote logins by OpenSSSH server.</td>
</tr>
<tr>
<td>sssd.service</td>
<td>yes</td>
<td>Disable <strong>sssd.service</strong> only if there are no users who log in the system over the network (for example by using LDAP or Kerberos). Red Hat recommends to disable all <strong>sssd-</strong>* units if you disable <strong>sssd.service</strong>.</td>
</tr>
<tr>
<td>syslog.service</td>
<td>yes</td>
<td>An alias for <strong>rsyslog.service</strong></td>
</tr>
<tr>
<td>tuned.service</td>
<td>yes</td>
<td>Disable <strong>tuned.service</strong> only if you do need to use performance tuning.</td>
</tr>
<tr>
<td>lvm2-lvmpolld.socket</td>
<td>yes</td>
<td>Disable <strong>lvm2-lvmpolld.socket</strong> only if you do not use Logical Volume Manager (LVM).</td>
</tr>
<tr>
<td>dnf-makecache.timer</td>
<td>yes</td>
<td>Disable <strong>dnf-makecache.timer</strong> only if you do not need your package metadata to be updated automatically.</td>
</tr>
<tr>
<td>unbound-anchor.timer</td>
<td>yes</td>
<td>Disable <strong>unbound-anchor.timer</strong> only if you do not need daily update of the root trust anchor for DNS Security Extensions (DNSSEC). This root trust anchor is used by Unbound resolver and resolver library for DNSSEC validation.</td>
</tr>
</tbody>
</table>

To find more information about a service, you can run one of the following commands:

```
~$ systemctl cat <service_name>
~$ systemctl help <service_name>
```

To find more information about a service, you can run one of the following commands:
The `systemctl cat` command provides the content of the service file located under 
`/usr/lib/systemd/system/<service>`, as well as all applicable overrides. The applicable overrides include unit file overrides from the `/etc/systemd/system/<service>` file or drop-in files from a corresponding `unit.type.d` directory.

For more information on drop-in files, see the `systemd.unit` man page.

The `systemctl help` command shows the man page of the particular service.

### 3.7. ADDITIONAL RESOURCES

For more information on systemd and its usage on Red Hat Enterprise Linux, see the resources listed below.

#### 3.7.1. Installed Documentation

- `systemctl(1)` – The manual page for the `systemctl` command line utility provides a complete list of supported options and commands.

- `systemd(1)` – The manual page for the `systemd` system and service manager provides more information about its concepts and documents available command line options and environment variables, supported configuration files and directories, recognized signals, and available kernel options.

- `systemd-delta(1)` – The manual page for the `systemd-delta` utility that allows to find extended and overridden configuration files.

- `systemd.directives(7)` – The manual page named `systemd.directives` provides detailed information about systemd directives.

- `systemd.unit(5)` – The manual page named `systemd.unit` provides detailed information about systemd unit files and documents all available configuration options.

- `systemd.service(5)` – The manual page named `systemd.service` documents the format of service unit files.

- `systemd.target(5)` – The manual page named `systemd.target` documents the format of target unit files.

- `systemd.kill(5)` – The manual page named `systemd.kill` documents the configuration of the process killing procedure.

#### 3.7.2. Online Documentation

- `systemd Home Page` – The project home page provides more information about systemd.
CHAPTER 4. MANAGING USER AND GROUP ACCOUNTS AND SETTING PERMISSIONS ON FILES

The control of users and groups is a core element of Red Hat Enterprise Linux system administration. This section explains how to add, manage, and delete users and groups in the graphical user interface and on the command line, and covers advanced topics, such as creating group directories.

4.1. INTRODUCTION TO USERS AND GROUPS

While users can be either people (meaning accounts tied to physical users) or accounts that exist for specific applications to use, groups are logical expressions of organization, tying users together for a common purpose. Users within a group share the same permissions to read, write, or execute files owned by that group.

Each user is associated with a unique numerical identification number called a user ID (UID). Likewise, each group is associated with a group ID (GID). A user who creates a file is also the owner and group owner of that file. The file is assigned separate read, write, and execute permissions for the owner, the group, and everyone else. The file owner can be changed only by root, and access permissions can be changed by both the root user and file owner.

4.2. RESERVED USER AND GROUP IDS

Red Hat Enterprise Linux reserves user and group IDs below 1000 for system users and groups. By default, the User Manager does not display the system users. Reserved user and group IDs are documented in the setup package. To view the documentation, use this command:

```
cat /usr/share/doc/setup*/uidgid
```

The recommended practice is to assign IDs starting at 5,000 that were not already reserved, as the reserved range can increase in the future. To make the IDs assigned to new users by default start at 5,000, change the UID_MIN and GID_MIN directives in the /etc/login.defs file:

```
UID_MIN                  5000
GID_MIN                  5000
```

**NOTE**

For users created before you changed UID_MIN and GID_MIN directives, UIDs will still start at the default 1000.

Even with new user and group IDs beginning with 5,000, it is recommended not to raise IDs reserved by the system above 1000 to avoid conflict with systems that retain the 1000 limit.

4.3. USER PRIVATE GROUPS

Red Hat Enterprise Linux uses a user private group (UPG) scheme, which makes UNIX groups easier to manage. A user private group is created whenever a new user is added to the system. It has the same name as the user for which it was created and that user is the only member of the user private group.
User private groups make it safe to set default permissions for a newly created file or directory, allowing both the user and the group of that user to make modifications to the file or directory.

The setting which determines what permissions are applied to a newly created file or directory is called a umask and is configured in the /etc/bashrc file. Traditionally on UNIX-based systems, the umask is set to 022, which allows only the user who created the file or directory to make modifications. Under this scheme, all other users, including members of the creator’s group, are not allowed to make any modifications. However, under the UPG scheme, this “group protection” is not necessary since every user has their own private group.

A list of all groups is stored in the /etc/group configuration file.

4.4. SHADOW PASSWORDS

In environments with multiple users, it is very important to use shadow passwords provided by the shadow-utils package to enhance the security of system authentication files. For this reason, the installation program enables shadow passwords by default.

The advantages of shadow passwords over the traditional way of storing passwords on UNIX-based systems are:

- Shadow passwords improve system security by moving encrypted password hashes from the world-readable /etc/passwd file to /etc/shadow, which is readable only by the root user.
- Shadow passwords store information about password aging.
- Shadow passwords allow to enforce some of the security policies set in the /etc/login.defs file.

Most utilities provided by the shadow-utils package work properly whether or not shadow passwords are enabled. However, since password aging information is stored exclusively in the /etc/shadow file, some utilities and commands do not work without first enabling shadow passwords:

- The chage utility for setting password aging parameters.
- The gpasswd utility for administrating the /etc/group file.
- The usermod command with the -e, --expiredate or -f, --inactive option.
- The useradd command with the -e, --expiredate or -f, --inactive option.

4.5. MANAGING USERS IN A GRAPHICAL ENVIRONMENT

The Users utility allows you to view, modify, add, and delete local users in the graphical user interface.

4.5.1. Using the Users Settings tool

Press the Super key to enter the Activities Overview, type Users and then press Enter. The Users settings tool appears. The Super key appears in a variety of options, depending on the keyboard and other hardware, but often as either the Windows or Command key, and typically to the left of the Space bar. Alternatively, you can open the Users utility from the Settings menu after clicking your user name in the top right corner of the screen.

To make changes to the user accounts, first select the Unlock button and authenticate yourself as indicated by the dialog box that appears. Note that unless you have superuser privileges, the application will prompt you to authenticate as root. To add and remove users, select the + and - button respectively.
To add a user to the administrative group wheel, change the Account Type from Standard to Administrator. To edit a user’s language setting, select the language and a drop-down menu appears.

Figure 4.1. The Users Settings Tool

When a new user is created, the account is disabled until a password is set. The Password drop-down menu, shown in Figure 4.2, “The Password Menu”, contains the options to set a password by the administrator immediately, choose a password by the user at the first login, or create a guest account with no password required to log in. You can also disable or enable an account from this menu.
4.6. MANAGING USERS USING COMMAND-LINE TOOLS

Apart from the Users settings tool described in Section 4.5.1, “Using the Users Settings tool”, which is designed for basic managing of users, you can use command line tools for managing users and groups that are listed in the table below.

Table 4.1. Command line utilities for managing users and groups

<table>
<thead>
<tr>
<th>Utilities</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>id</td>
<td>Displays user and group IDs.</td>
</tr>
<tr>
<td>useradd, usermod, userdel</td>
<td>Standard utilities for adding, modifying, and deleting user accounts.</td>
</tr>
<tr>
<td>groupadd, groupmod, groupdel</td>
<td>Standard utilities for adding, modifying, and deleting groups.</td>
</tr>
<tr>
<td>gpasswd</td>
<td>Utility primarily used for modification of group password in the /etc/gshadow file which is used by the newgrp command.</td>
</tr>
<tr>
<td>pwck, grpck</td>
<td>Utilities that can be used for verification of the password, group, and associated shadow files.</td>
</tr>
</tbody>
</table>
Utilities that can be used for the conversion of passwords to shadow passwords, or back from shadow passwords to standard passwords.

Utilities that can be used for conversion of shadowed information for group accounts.

<table>
<thead>
<tr>
<th>Utilities</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pwconv, pwunconv</td>
<td>Utilities that can be used for the conversion of passwords to shadow passwords, or back from shadow passwords to standard passwords.</td>
</tr>
<tr>
<td>grpconv, grpunconv</td>
<td>Similar to the previous, these utilities can be used for conversion of shadowed information for group accounts.</td>
</tr>
</tbody>
</table>

4.6.1. Adding a new user

To add a new user to the system, type the following at a shell prompt as root:

```
useradd options username
```

Here `options` are command-line options as described in Table 4.2, “Common useradd command-line options”.

**WARNING**

With RHEL 8, you cannot use all-numeric user names. The reason for not allowing such names is that this can confuse tools that work with user names and user ids, which are numbers.

By default, the `useradd` command creates a locked user account. To unlock the account, run the following command as root to assign a password:

```
passwd username
```

Table 4.2. Common useradd command-line options

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>-c 'comment'</code></td>
<td><code>comment</code> can be replaced with any string. This option is generally used to specify the full name of a user.</td>
</tr>
<tr>
<td><code>-d home_directory</code></td>
<td>Home directory to be used instead of default <code>/home/username/</code>.</td>
</tr>
<tr>
<td><code>-e date</code></td>
<td>Date for the account to be disabled in the format YYYY-MM-DD.</td>
</tr>
</tbody>
</table>
Option

-`f days`  
Number of days after the password expires until the account is disabled. If 0 is specified, the account is disabled immediately after the password expires. If -1 is specified, the account is not disabled after the password expires.

-`g` group_name  
Group name or group number for the user’s default (primary) group. The group must exist prior to being specified here.

-`G` group_list  
List of additional (supplementary, other than default) group names or group numbers, separated by commas, of which the user is a member. The groups must exist prior to being specified here.

-`m`  
Create the home directory if it does not exist.

-`M`  
Do not create the home directory.

-`N`  
Do not create a user private group for the user.

-`p` password  
The password encrypted with crypt.

-`r`  
Create a system account with a UID less than 1000 and without a home directory.

-`s`  
User’s login shell, which defaults to /bin/bash.

-`u` uid  
User ID for the user, which must be unique and greater than 999.

**IMPORTANT**

The default range of IDs for system and normal users has been changed in Red Hat Enterprise Linux 7 from earlier releases. Before Red Hat Enterprise Linux 7, UID 1-499 was used for system users and values above for normal users. The default range for system users is now 1-999. This change might cause problems when migrating to Red Hat Enterprise Linux 8 with existing users having UIDs and GIDs between 500 and 999. The default ranges of UID and GID can be changed in the `/etc/login.defs` file.

The following steps illustrate what happens if the command `useradd juan` is issued on a system that has shadow passwords enabled:

1. A new line for `juan` is created in `/etc/passwd`:
   
   ```
   juan:x:1001:1001::/home/juan:/bin/bash
   ```

   The line has the following characteristics:
• It begins with the user name **juan**.

• There is an **x** for the password field indicating that the system is using shadow passwords.

• A UID greater than 999 is created. Under Red Hat Enterprise Linux 7, UIDs below 1000 are reserved for system use and should not be assigned to users.

• A GID greater than 999 is created. Under Red Hat Enterprise Linux 7, GIDs below 1000 are reserved for system use and should not be assigned to users.

• The optional **GECOS** information is left blank. The GECOS field can be used to provide additional information about the user, such as their full name or phone number.

• The home directory for **juan** is set to `/home/juan/`.

• The default shell is set to `/bin/bash`.

2. A new line for **juan** is created in `/etc/shadow`:

```
juan:!!:14798:0:99999:7:::
```

The line has the following characteristics:

• It begins with the user name **juan**.

• Two exclamation marks (**!!**) appear in the password field of the `/etc/shadow` file, which locks the account.

```
NOTE
If an encrypted password is passed using the **-p** flag, it is placed in the `/etc/shadow` file on the new line for the user.
```

• The password is set to never expire.

3. A new line for a group named **juan** is created in `/etc/group`:

```
juan:x:1001:
```

A group with the same name as a user is called a **user private group**. For more information on user private groups, see Section 4.3, "User private groups".

The line created in `/etc/group` has the following characteristics:

• It begins with the group name **juan**.

• An **x** appears in the password field indicating that the system is using shadow group passwords.

• The GID matches the one listed for **juan**'s primary group in `/etc/passwd`.

4. A new line for a group named **juan** is created in `/etc/gshadow`:

```
juan:!::
```

The line has the following characteristics:
• It begins with the group name `juan`.
• An exclamation mark (!) appears in the password field of the `/etc/gshadow` file, which locks the group.
• All other fields are blank.

5. A directory for user `juan` is created in the `/home` directory:

```
~]# ls -ld /home/juan
drwx------. 4 juan juan 4096 Mar  3 18:23 /home/juan
```

This directory is owned by user `juan` and group `juan`. It has `read`, `write`, and `execute` privileges only for the user `juan`. All other permissions are denied.

6. The files within the `/etc/skel/` directory (which contain default user settings) are copied into the new `/home/juan/` directory:

```
~]# ls -la /home/juan
```

At this point, a locked account called `juan` exists on the system. To activate it, the administrator must next assign a password to the account using the `passwd` command and, optionally, set password aging guidelines.

### 4.6.2. Adding a new group

To add a new group to the system, type the following at a shell prompt as `root`:

```
groupadd options group_name
```

Here `options` are command-line options as described in Table 4.3, “Common groupadd command-line options”.

**WARNING**

With RHEL 8, you cannot use all-numeric group names. The reason for not allowing such names is that this can confuse tools that work with group names and group ids, which are numbers.

Table 4.3. Common groupadd command-line options
<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-f, --force</td>
<td>When used with -g \textit{gid} and \textit{gid} already exists, \texttt{groupadd} will choose another unique \textit{gid} for the group.</td>
</tr>
<tr>
<td>-g \textit{gid}</td>
<td>Group ID for the group, which must be unique and greater than 999.</td>
</tr>
<tr>
<td>-K, --key \textit{key}=\textit{value}</td>
<td>Override \texttt{/etc/login.defs} defaults.</td>
</tr>
<tr>
<td>-o, --non-unique</td>
<td>Allows creating groups with duplicate GID.</td>
</tr>
<tr>
<td>-p, --password \textit{password}</td>
<td>Use this encrypted password for the new group.</td>
</tr>
<tr>
<td>-r</td>
<td>Create a system group with a \textit{GID} less than 1000.</td>
</tr>
</tbody>
</table>

4.6.3. Adding an existing user to an existing group

Use the \texttt{usermod} utility to add an already existing user to an already existing group.

Various options of \texttt{usermod} have different impact on user’s primary group and on his or her supplementary groups.

To override user’s primary group, run the following command as \texttt{root}:

\begin{verbatim}
usermod -g \textit{group\_name} \textit{user\_name}
\end{verbatim}

To override user’s supplementary groups, run the following command as \texttt{root}:

\begin{verbatim}
usermod -G \textit{group\_name1\_group\_name2,...\ user\_name}
\end{verbatim}

Note that in this case all previous supplementary groups of the user are replaced by the new group or several new groups.

To add one or more groups to user’s supplementary groups, run one of the following commands as \texttt{root}:

\begin{verbatim}
usermod -aG \textit{group\_name1\_group\_name2,...\ user\_name}
usermod --append -G \textit{group\_name1\_group\_name2,...\ user\_name}
\end{verbatim}

Note that in this case the new group is added to user’s current supplementary groups.

4.6.4. Creating group directories

System administrators usually like to create a group for each major project and assign people to the group when they need to access that project’s files. With this traditional scheme, file management is difficult; when someone creates a file, it is associated with the primary group to which they belong. When a single person works on multiple projects, it becomes difficult to associate the right files with the right group. However, with the UPG scheme, groups are automatically assigned to files created within a
directory with the setgid bit set. The setgid bit makes managing group projects that share a common directory very simple because any files a user creates within the directory are owned by the group that owns the directory.

For example, a group of people need to work on files in the /opt/myproject/ directory. Some people are trusted to modify the contents of this directory, but not everyone.

1. As root, create the /opt/myproject/ directory by typing the following at a shell prompt:

   ```bash
   mkdir /opt/myproject
   ```

2. Add the myproject group to the system:

   ```bash
   groupadd myproject
   ```

3. Associate the contents of the /opt/myproject/ directory with the myproject group:

   ```bash
   chown root:myproject /opt/myproject
   ```

4. Allow users in the group to create files within the directory and set the setgid bit:

   ```bash
   chmod 2775 /opt/myproject
   ```

   At this point, all members of the myproject group can create and edit files in the /opt/myproject/ directory without the administrator having to change file permissions every time users write new files. To verify that the permissions have been set correctly, run the following command:

   ```bash
   ls -ld /opt/myproject
   ```

   ```bash
   drwxrwsr-x. 3 root myproject 4096 Mar  3 18:31 /opt/myproject
   ```

5. Add users to the myproject group:

   ```bash
   usermod -aG myproject username
   ```

4.6.5. Setting default permissions for new files using umask

When a process creates a file, the file has certain default permissions, for example, -rw-rw-r-. These initial permissions are partially defined by the file mode creation mask, also called file permission mask or umask. Every process has its own umask, for example, bash has umask 0022 by default. Process umask can be changed.

4.6.5.1. What umask consists of

A umask consists of bits corresponding to standard file permissions. For example, for umask 0137, the digits mean that:

- 0 = no meaning, it is always 0 (umask does not affect special bits)
- 1 = for owner permissions, the execute bit is set
- 3 = for group permissions, the execute and write bits are set
Umasks can be represented in binary, octal, or symbolic notation. For example, the octal representation 0137 equals symbolic representation u=rw-,g=r--,o=---. Symbolic notation specification is the reverse of the octal notation specification: it shows the allowed permissions, not the prohibited permissions.

4.6.5.2. How umask works

**Umask** prohibits permissions from being set for a file:

- When a bit is set in umask, it is unset in the file.
- When a bit is not set in umask, it can be set in the file, depending on other factors.

The following figure shows how umask 0137 affects creating a new file.

**Figure 4.3. Applying umask when creating a file**

![Diagram showing how umask affects permissions](image)

**IMPORTANT**

For security reasons, a regular file cannot have execute permissions by default. Therefore, even if umask is 0000, which does not prohibit any permissions, a new regular file still does not have execute permissions. However, directories can be created with execute permissions:

```
[john@server tmp]$ umask 0000
[john@server tmp]$ touch file
[john@server tmp]$ mkdir directory
[john@server tmp]$ ls -lh .
total 0
drwxrwxrwx. 2 john john 40 Nov 2 13:17 directory
-rw-rw-rw-. 1 john john 0 Nov 2 13:17 file
```
4.6.5.3. Managing umask in Shells

For popular shells, such as **bash**, **ksh**, **zsh** and **tcsh**, **umask** is managed using the **umask** shell builtin. Processes started from shell inherit its umask.

4.6.5.3.1. Displaying the current mask

To show the current **umask** in octal notation:

```
~]$ umask
0022
```

To show the current **umask** in symbolic notation:

```
umask -S
u=rwx,g=rx,o=rx
```

4.6.5.3.2. Setting mask in shell using umask

To set **umask** for the current shell session using octal notation run:

```
umask octal_mask
```

Substitute **octal_mask** with four or less digits from **0** to **7**. When three or less digits are provided, permissions are set as if the command contained leading zeros. For example, **umask 7** translates to **0007**.

**Example 4.1. Setting umask Using Octal Notation**

To prohibit new files from having write and execute permissions for owner and group, and from having any permissions for others:

```
umask 0337
```

Or:

```
umask 337
```

To set **umask** for the current shell session using symbolic notation:

```
umask -S symbolic_mask
```

**Example 4.2. Setting umask Using Symbolic Notation**

To set **umask 0337** using symbolic notation:

```
umask -S u=r,g=r,o=
```

4.6.5.3.3. Working with the default shell umask
Shells usually have a configuration file where their default `umask` is set. For `bash`, it is `/etc/bashrc`. To show the default `bash` umask:

```bash
grep -i -B 1 umask /etc/bashrc
```

The output shows if `umask` is set, either using the `umask` command or the `UMASK` variable. In the following example, `umask` is set to `022` using the `umask` command:

```bash
grep -i -B 1 umask /etc/bashrc
# By default, we want umask to get set. This sets it for non-login shell. — if [ $UID -gt 199 ] && [ "id -gn" = "id -un" ]; then
  umask 002
else
  umask 022
```

To change the default `umask` for `bash`, change the `umask` command call or the `UMASK` variable assignment in `/etc/bashrc`. This example changes the default `umask` to `0227`:

```bash
if [ $UID -gt 199 ] && [ "id -gn" = "id -un" ]; then
  umask 002
else
  umask 227
```

### 4.6.5.3.4. Working with the default shell umask of a specific user

By default, `bash` `umask` of a new user defaults to the one defined in `/etc/bashrc`.

To change `bash` `umask` for a particular user, add a call to the `umask` command in `$HOME/.bashrc` file of that user. For example, to change `bash umask` of user `john` to `0227`:

```bash
john@server ~]$ echo 'umask 227' >> /home/john/.bashrc
```

### 4.6.5.3.5. Setting default permissions for newly created home directories

To change permissions with which user home directories are created, change the `UMASK` variable in the `/etc/login.defs` file:

```bash
# The permission mask is initialized to this value. If not specified,
# the permission mask will be initialized to 022.
UMASK 077
```

### 4.7. ADDITIONAL RESOURCES

For more information on how to manage users and groups on Red Hat Enterprise Linux, see the resources listed below.

#### 4.7.1. Installed Documentation

For information about various utilities for managing users and groups, see the following manual pages:

- `useradd(8)` — The manual page for the `useradd` command documents how to use it to create new users.
• `userdel(8)` — The manual page for the `userdel` command documents how to use it to delete users.

• `usermod(8)` — The manual page for the `usermod` command documents how to use it to modify users.

• `groupadd(8)` — The manual page for the `groupadd` command documents how to use it to create new groups.

• `groupdel(8)` — The manual page for the `groupdel` command documents how to use it to delete groups.

• `groupmod(8)` — The manual page for the `groupmod` command documents how to use it to modify group membership.

• `gpasswd(1)` — The manual page for the `gpasswd` command documents how to manage the `/etc/group` file.

• `grpck(8)` — The manual page for the `grpck` command documents how to use it to verify the integrity of the `/etc/group` file.

• `pwck(8)` — The manual page for the `pwck` command documents how to use it to verify the integrity of the `/etc/passwd` and `/etc/shadow` files.

• `pwconv(8)` — The manual page for the `pwconv`, `pwunconv`, `grpconv`, and `grpunconv` commands documents how to convert shadowed information for passwords and groups.

• `id(1)` — The manual page for the `id` command documents how to display user and group IDs.

For information about related configuration files, see:

• `group(5)` — The manual page for the `/etc/group` file documents how to use this file to define system groups.

• `passwd(5)` — The manual page for the `/etc/passwd` file documents how to use this file to define user information.

• `shadow(5)` — The manual page for the `/etc/shadow` file documents how to use this file to set passwords and account expiration information for the system.
CHAPTER 5. USING THE CHRONY SUITE TO CONFIGURE NTP

5.1. INTRODUCTION TO CONFIGURING NTP WITH CHRONY

Accurate timekeeping is important for a number of reasons in IT. In networking for example, accurate time stamps in packets and logs are required. In Linux systems, the NTP protocol is implemented by a daemon running in user space.

The user space daemon updates the system clock running in the kernel. The system clock can keep time by using various clock sources. Usually, the Time Stamp Counter (TSC) is used. The TSC is a CPU register which counts the number of cycles since it was last reset. It is very fast, has a high resolution, and there are no interruptions.

In Red Hat Enterprise Linux 8, the NTP protocol is implemented by the chronyd daemon, available from the repositories in the chrony package.

These sections describe the use of the chrony suite.

5.2. INTRODUCTION TO CHRONY SUITE

chrony is an implementation of the Network Time Protocol (NTP). You can use chrony:

- To synchronize the system clock with NTP servers
- To synchronize the system clock with a reference clock, for example a GPS receiver
- To synchronize the system clock with a manual time input
- As an NTPv4 (RFC 5905) server or peer to provide a time service to other computers in the network

chrony performs well in a wide range of conditions, including intermittent network connections, heavily congested networks, changing temperatures (ordinary computer clocks are sensitive to temperature), and systems that do not run continuously, or run on a virtual machine.

Typical accuracy between two machines synchronized over the Internet is within a few milliseconds, and for machines on a LAN within tens of microseconds. Hardware timestamping or a hardware reference clock may improve accuracy between two machines synchronized to a sub-microsecond level.

chrony consists of chronyd, a daemon that runs in user space, and chronyc, a command line program which can be used to monitor the performance of chronyd and to change various operating parameters when it is running.

The chrony daemon, chronyd, can be monitored and controlled by the command line utility chronyc. This utility provides a command prompt which allows entering a number of commands to query the current state of chronyd and make changes to its configuration. By default, chronyd accepts only commands from a local instance of chronyc, but it can be configured to accept monitoring commands also from remote hosts. The remote access should be restricted.

5.2.1. Using chronyc to control chronyd

To make changes to the local instance of chronyd using the command line utility chronyc in interactive mode, enter the following command as root:

```
~]$ chronyc
```
chronyc must run as root if some of the restricted commands are to be used.

The chronyc command prompt will be displayed as follows:

```
chronyc>
```

You can type help to list all of the commands.

The utility can also be invoked in non-interactive command mode if called together with a command as follows:

```
chronyc command
```

**NOTE**

Changes made using chronyc are not permanent, they will be lost after a chronyd restart. For permanent changes, modify /etc/chrony.conf.

### 5.3. DIFFERENCES BETWEEN CHRONY AND NTP

**Network Time Protocol (NTP)** has two different implementations with similar basic functionality - ntp and chrony.

Both ntp and chrony can operate as an NTP client in order to synchronize the system clock with NTP servers and they can operate as an NTP server for other computers in the network. Each implementation has some unique features. For comparison of ntp and chrony, see Comparison of NTP implementations.

Configuration specific to an NTP client is identical in most cases. NTP servers are specified with the server directive. A pool of servers can be specified with the pool directive.

Configuration specific to an NTP server differs in how the client access is controlled. By default, ntpd responds to client requests from any address. The access can be restricted with the restrict directive, but it is not possible to disable the access completely if ntpd uses any servers as a client. chronyd allows no access by default and operates as an NTP client only. To make chrony operate as an NTP server, you need to specify some addresses within the allow directive.

ntpd and chronyd differ also in the default behavior with respect to corrections of the system clock. ntpd corrects the clock by step when the offset is larger than 128 milliseconds. If the offset is larger than 1000 seconds, ntpd exits unless it is the first correction of the clock and ntpd is started with the -g option. chronyd does not step the clock by default, but the default chrony.conf file provided in the chrony package allows steps in the first three updates of the clock. After that, all corrections are made slowly by speeding up or slowing down the clock. The chronyc makestep command can be issued to force chronyd to step the clock at any time.

### 5.4. MIGRATING TO CHRONY

In Red Hat Enterprise Linux 7, users could choose between ntp and chrony to ensure accurate timekeeping. For differences between ntp and chrony, ntpd and chronyd, see Differences between ntpd and chronyd.

In Red Hat Enterprise Linux 8, ntp is no longer supported. chrony is enabled by default. For this reason, you might need to migrate from ntp to chrony.
Migrating from ntp to chrony is straightforward in most cases. The corresponding names of the programs, configuration files and services are:

<table>
<thead>
<tr>
<th>ntp name</th>
<th>chrony name</th>
</tr>
</thead>
<tbody>
<tr>
<td>/etc/ntp.conf</td>
<td>/etc/chrony.conf</td>
</tr>
<tr>
<td>/etc/ntp/keys</td>
<td>/etc/chrony.keys</td>
</tr>
<tr>
<td>ntpd</td>
<td>chronyd</td>
</tr>
<tr>
<td>ntpq</td>
<td>chronyc</td>
</tr>
<tr>
<td>ntpd.service</td>
<td>chronyd.service</td>
</tr>
<tr>
<td>ntp-wait.service</td>
<td>chrony-wait.service</td>
</tr>
</tbody>
</table>

The ntpdate and sntp utilities, which are included in the ntp distribution, can be replaced with chronyd using the -q option or the -t option. The configuration can be specified on the command line to avoid reading /etc/chrony.conf. For example, instead of running ntpdate ntp.example.com, chronyd could be started as:

```
# chronyd -q 'server ntp.example.com iburst'
2018-05-18T12:37:43Z chronyd version 3.3 starting (+CMDMON +NTP +REFCLOCK +RTC +PRIVDROP +SCFILTER +SIGNAL +ASYNCDNS +SECHASH +IPV6 +DEBUG)
2018-05-18T12:37:48Z System clock wrong by 0.003159 seconds (step)
```

The ntpstat utility, which was previously included in the ntp package and supported only ntpd, now supports both ntpd and chronyd. It is available in the ntpstat package.

5.4.1. Migration script

A Python script called ntp2chrony.py is included in the documentation of the chrony package (/usr/share/doc/chrony). The script automatically converts an existing ntp configuration to chrony. It supports the most common directives and options in the ntp.conf file. Any lines that are ignored in the conversion are included as comments in the generated chrony.conf file for review. Keys that are specified in the ntp key file, but are not marked as trusted keys in ntp.conf are included in the generated chrony.keys file as comments.

By default, the script does not overwrite any files. If /etc/chrony.conf or /etc/chrony.keys already exist, the -b option can be used to rename the file as a backup. The script supports other options. The --help option prints all supported options.

An example of an invocation of the script with the default ntp.conf provided in the ntp package is:

```
# python3 /usr/share/doc/chrony/ntp2chrony.py -b -v
Reading /etc/ntp.conf
```
Reading /etc/ntp/crypto/pw
Reading /etc/ntp/keys
Writing /etc/chrony.conf
Writing /etc/chrony.keys

The only directive ignored in this case is disable monitor, which has a chrony equivalent in the noclientlog directive, but it was included in the default ntp.conf only to mitigate an amplification attack.

The generated chrony.conf file typically includes a number of allow directives corresponding to the restrict lines in ntp.conf. If you do not want to run chronyd as an NTP server, remove all allow directives from chrony.conf.

5.4.2. Timesync role

Note that using the timesync role on your Red Hat Enterprise Linux 7 system facilitates the migration to chrony, because you can use the same playbook on all versions of RHEL starting with RHEL 6 regardless of whether the system uses ntp or chrony to implement the NTP protocol.

For more information on using the timesync role to manage time synchronization, see System roles documentation.

5.5. CONFIGURING CHRONY

The default configuration file for chronyd is /etc/chrony.conf. The -f option can be used to specify an alternate configuration file path. See the chrony.conf(5) man page for further options. For a complete list of the directives that can be used see The chronyd configuration file.

Below is a selection of chronyd configuration options:

Comments
Comments should be preceded by #, %, ; or !

allow
Optionally specify a host, subnet, or network from which to allow NTP connections to a machine acting as NTP server. The default is not to allow connections.

Examples:

allow 192.0.2.0/24
Use this command to grant access to a specific network.

allow 2001:0db8:85a3::8a2e:0370:7334
Use this this command to grant access to an IPv6.

The UDP port number 123 needs to be open in the firewall in order to allow the client access:

~]# firewall-cmd --zone=public --add-port=123/udp
If you want to open port 123 permanently, use the --permanent option:

~]# firewall-cmd --permanent --zone=public --add-port=123/udp
cmdallow

This is similar to the allow directive (see section allow), except that it allows control access (rather than NTP client access) to a particular subnet or host. (By “control access” is meant that chronyc can be run on those hosts and successfully connect to chronyd on this computer.) The syntax is identical. There is also a cmddeny all directive with similar behavior to the cmdallow all directive.

dumpdir

Path to the directory to save the measurement history across restarts of chronyd (assuming no changes are made to the system clock behavior whilst it is not running). If this capability is to be used (via the dumponexit command in the configuration file, or the dump command in chronyc), the dumpdir command should be used to define the directory where the measurement histories are saved.

dumponexit

If this command is present, it indicates that chronyd should save the measurement history for each of its time sources recorded whenever the program exits. (See the dumpdir command above).

hwtimestamp

The hwtimestamp directive enables hardware timestamping for extremely accurate synchronization. For more details, see the chrony.conf(5) manual page.

local

The local keyword is used to allow chronyd to appear synchronized to real time from the viewpoint of clients polling it, even if it has no current synchronization source. This option is normally used on the “master” computer in an isolated network, where several computers are required to synchronize to one another, and the “master” is kept in line with real time by manual input.

An example of the command is:

```
local stratum 10
```

A large value of 10 indicates that the clock is so many hops away from a reference clock that its time is unreliable. If the computer ever has access to another computer which is ultimately synchronized to a reference clock, it will almost certainly be at a stratum less than 10. Therefore, the choice of a high value like 10 for the local command prevents the machine’s own time from ever being confused with real time, were it ever to leak out to clients that have visibility of real servers.

log

The log command indicates that certain information is to be logged. It accepts the following options:

measurements

This option logs the raw NTP measurements and related information to a file called measurements.log.

statistics

This option logs information about the regression processing to a file called statistics.log.

tracking

This option logs changes to the estimate of the system’s gain or loss rate, and any slews made, to a file called tracking.log.

rtc

This option logs information about the system’s real-time clock.

refclocks

This option logs the raw and filtered reference clock measurements to a file called refclocks.log.

tempcomp
This option logs the temperature measurements and system rate compensations to a file called `tempcomp.log`. The log files are written to the directory specified by the `logdir` command.

An example of the command is:

```
log measurements statistics tracking
```

**logdir**

This directive allows the directory where log files are written to be specified. An example of the use of this directive is:

```
logdir /var/log/chrony
```

**makestep**

Normally `chronyd` will cause the system to gradually correct any time offset, by slowing down or speeding up the clock as required. In certain situations, the system clock may be so far adrift that this slewing process would take a very long time to correct the system clock. This directive forces `chronyd` to step system clock if the adjustment is larger than a threshold value, but only if there were no more clock updates since `chronyd` was started than a specified limit (a negative value can be used to disable the limit). This is particularly useful when using reference clock, because the `initstepslew` directive only works with NTP sources. An example of the use of this directive is:

```
makestep 1000 10
```

This would step the system clock if the adjustment is larger than 1000 seconds, but only in the first ten clock updates.

**maxchange**

This directive sets the maximum allowed offset corrected on a clock update. The check is performed only after the specified number of updates to allow a large initial adjustment of the system clock. When an offset larger than the specified maximum occurs, it will be ignored for the specified number of times and then `chronyd` will give up and exit (a negative value can be used to never exit). In both cases a message is sent to syslog. An example of the use of this directive is:

```
maxchange 1000 1 2
```

After the first clock update, `chronyd` will check the offset on every clock update, it will ignore two adjustments larger than 1000 seconds and exit on another one.

**maxupdateskew**

One of `chronyd`'s tasks is to work out how fast or slow the computer’s clock runs relative to its reference sources. In addition, it computes an estimate of the error bounds around the estimated value. If the range of error is too large, it indicates that the measurements have not settled down yet, and that the estimated gain or loss rate is not very reliable.

The `maxupdateskew` parameter is the threshold for determining whether an estimate is too unreliable to be used. By default, the threshold is 1000 ppm.
The format of the syntax is:

```
maxupdateskew skew-in-ppm
```

Typical values for `skew-in-ppm` might be 100 for a dial-up connection to servers over a telephone line, and 5 or 10 for a computer on a LAN.

It should be noted that this is not the only means of protection against using unreliable estimates. At all times, `chronyd` keeps track of both the estimated gain or loss rate, and the error bound on the estimate. When a new estimate is generated following another measurement from one of the sources, a weighted combination algorithm is used to update the master estimate. So if `chronyd` has an existing highly-reliable master estimate and a new estimate is generated which has large error bounds, the existing master estimate will dominate in the new master estimate.

**minsources**

The `minsources` directive sets the minimum number of sources that need to be considered as selectable in the source selection algorithm before the local clock is updated.

The format of the syntax is:

```
minsources number-of-sources
```

By default, `number-of-sources` is 1. Setting `minsources` to a larger number can be used to improve the reliability, because multiple sources will need to correspond with each other.

**noclientlog**

This directive, which takes no arguments, specifies that client accesses are not to be logged. Normally they are logged, allowing statistics to be reported using the clients command in `chronyc` and enabling the clients to use interleaved mode with the `xleave` option in the `server` directive.

**reselectdist**

When `chronyd` selects synchronization source from available sources, it will prefer the one with minimum synchronization distance. However, to avoid frequent reselecting when there are sources with similar distance, a fixed distance is added to the distance for sources that are currently not selected. This can be set with the `reselectdist` option. By default, the distance is 100 microseconds.

The format of the syntax is:

```
reselectdist dist-in-seconds
```

**stratumweight**

The `stratumweight` directive sets how much distance should be added per stratum to the synchronization distance when `chronyd` selects the synchronization source from available sources.

The format of the syntax is:

```
stratumweight dist-in-seconds
```

By default, `dist-in-seconds` is 1 millisecond. This means that sources with lower stratum are usually preferred to sources with higher stratum even when their distance is significantly worse. Setting `stratumweight` to 0 makes `chronyd` ignore stratum when selecting the source.

**rtcfile**

The `rtcfile` directive defines the name of the file in which `chronyd` can save parameters associated with tracking the accuracy of the system’s real-time clock (RTC).
The format of the syntax is:

```
  rtcfile /var/lib/chrony/rtc
```

**chronyd** saves information in this file when it exits and when the **writertc** command is issued in **chronyc**. The information saved is the RTC’s error at some epoch, that epoch (in seconds since January 1 1970), and the rate at which the RTC gains or loses time. Not all real-time clocks are supported as their code is system-specific. Note that if this directive is used then the real-time clock should not be manually adjusted as this would interfere with **chrony**’s need to measure the rate at which the real-time clock drifts if it was adjusted at random intervals.

**rtcsync**

The **rtcsync** directive is present in the **/etc/chrony.conf** file by default. This will inform the kernel the system clock is kept synchronized and the kernel will update the real-time clock every 11 minutes.

### 5.5.1. Configuring chrony for security

**chronyc** can access **chronyd** in two ways:

- Internet Protocol, IPv4 or IPv6.
- Unix domain socket, which is accessible locally by the **root** or **chrony** user.

By default, **chronyc** connects to the Unix domain socket. The default path is **/var/run/chrony/chronyd.sock**. If this connection fails, which can happen for example when **chronyc** is running under a non-root user, **chronyc** tries to connect to 127.0.0.1 and then ::1.

Only the following monitoring commands, which do not affect the behavior of **chronyd**, are allowed from the network:

- activity
- manual list
- rtcdarata
- smoothing
- sources
- sourcestats
- tracking
- waitsync

The set of hosts from which **chronyd** accepts these commands can be configured with the **cmdallow** directive in the configuration file of **chronyd**, or the **cmdallow** command in **chronyc**. By default, the commands are accepted only from localhost (127.0.0.1 or ::1).

All other commands are allowed only through the Unix domain socket. When sent over the network, **chronyd** responds with a **Not authorised** error, even if it is from localhost.

**Accessing chronyd remotely with chronyc**
1. Allow access from both IPv4 and IPv6 addresses by adding the following to the `/etc/chrony.conf` file:

   ```
   bindcmdaddress 0.0.0.0
   ```

   or

   ```
   bindcmdaddress :
   ```

2. Allow commands from the remote IP address, network, or subnet by using the `cmdallow` directive.
   Add the following content to the `/etc/chrony.conf` file:

   ```
   cmdallow 192.168.1.0/24
   ```

3. Open port 323 in the firewall to connect from a remote system.

   ```
   ~$ firewall-cmd --zone=public --add-port=323/udp
   ```

   If you want to open port 323 permanently, use the `--permanent`.

   ```
   ~$ firewall-cmd --permanent --zone=public --add-port=323/udp
   ```

Note that the `allow` directive is for NTP access whereas the `cmdallow` directive is to enable receiving of remote commands. It is possible to make these changes temporarily using `chronyc` running locally. Edit the configuration file to make permanent changes.

5.6. USING CHRONY

5.6.1. Installing chrony

The `chrony` suite is installed by default on Red Hat Enterprise Linux. To ensure that it is, run the following command as `root`:

```
~$ yum install chrony
```

The default location for the `chrony` daemon is `/usr/sbin/chronyd`. The command line utility will be installed to `/usr/bin/chronyc`.

5.6.2. Checking the status of chronyd

To check the status of `chronyd`, issue the following command:

```
~$ systemctl status chronyd
chronyd.service - NTP client/server
   Loaded: loaded (/usr/lib/systemd/system/chronyd.service; enabled)
   Active: active (running) since Wed 2013-06-12 22:23:16 CEST; 11h ago
```

5.6.3. Starting chronyd

To start `chronyd`, issue the following command as `root`:
5.6.4. Stopping chronyd

To stop chronyd, issue the following command as root:

```
~# systemctl stop chronyd
```

To prevent chronyd from starting automatically at system start, issue the following command as root:

```
~# systemctl disable chronyd
```

5.6.5. Checking if chrony is synchronized

To check if chrony is synchronized, make use of the tracking, sources, and sourcestats commands.

5.6.5.1. Checking chrony tracking

To check chrony tracking, issue the following command:

```
~$ chronyc tracking
Reference ID    : CB00710F (foo.example.net)
Stratum         : 3
Ref time (UTC)  : Fri Jan 27 09:49:17 2017
System time     :  0.000006523 seconds slow of NTP time
Last offset     : -0.000006747 seconds
RMS offset      : 0.000035822 seconds
Frequency       : 3.225 ppm slow
Residual freq   : 0.000 ppm
Skew            : 0.129 ppm
Root delay      : 0.013639022 seconds
Root dispersion : 0.001100737 seconds
Update interval : 64.2 seconds
Leap status     : Normal
```

The fields are as follows:

**Reference ID**

This is the reference ID and name (or IP address) if available, of the server to which the computer is currently synchronized. Reference ID is a hexadecimal number to avoid confusion with IPv4 addresses.

**Stratum**

The stratum indicates how many hops away from a computer with an attached reference clock we are. Such a computer is a stratum-1 computer, so the computer in the example is two hops away (that is to say, a.b.c is a stratum-2 and is synchronized from a stratum-1).

**Ref time**

This is the time (UTC) at which the last measurement from the reference source was processed.
System time

In normal operation, chronyd never steps the system clock, because any jump in the timescale can have adverse consequences for certain application programs. Instead, any error in the system clock is corrected by slightly speeding up or slowing down the system clock until the error has been removed, and then returning to the system clock’s normal speed. A consequence of this is that there will be a period when the system clock (as read by other programs using the gettimeofday() system call, or by the date command in the shell) will be different from chronyd’s estimate of the current true time (which it reports to NTP clients when it is operating in server mode). The value reported on this line is the difference due to this effect.

Last offset

This is the estimated local offset on the last clock update.

RMS offset

This is a long-term average of the offset value.

Frequency

The "frequency" is the rate by which the system’s clock would be wrong if chronyd was not correcting it. It is expressed in ppm (parts per million). For example, a value of 1 ppm would mean that when the system’s clock thinks it has advanced 1 second, it has actually advanced by 1.000001 seconds relative to true time.

Residual freq

This shows the "residual frequency" for the currently selected reference source. This reflects any difference between what the measurements from the reference source indicate the frequency should be and the frequency currently being used. The reason this is not always zero is that a smoothing procedure is applied to the frequency. Each time a measurement from the reference source is obtained and a new residual frequency computed, the estimated accuracy of this residual is compared with the estimated accuracy (see skew) of the existing frequency value. A weighted average is computed for the new frequency, with weights depending on these accuracies. If the measurements from the reference source follow a consistent trend, the residual will be driven to zero over time.

Skew

This is the estimated error bound on the frequency.

Root delay

This is the total of the network path delays to the stratum-1 computer from which the computer is ultimately synchronized. Root delay values are printed in nanosecond resolution. In certain extreme situations, this value can be negative. (This can arise in a symmetric peer arrangement where the computers’ frequencies are not tracking each other and the network delay is very short relative to the turn-around time at each computer.)

Root dispersion

This is the total dispersion accumulated through all the computers back to the stratum-1 computer from which the computer is ultimately synchronized. Dispersion is due to system clock resolution, statistical measurement variations etc. Root dispersion values are printed in nanosecond resolution.

Leap status

This is the leap status, which can be Normal, Insert second, Delete second or Not synchronized.

5.6.5.2. Checking chrony sources

The sources command displays information about the current time sources that chronyd is accessing.

The optional argument -v can be specified, meaning verbose. In this case, extra caption lines are shown as a reminder of the meanings of the columns.
### chronyc sources
```
-]$ chronyc sources
210 Number of sources = 3
MS Name/IP address Stratum Poll Reach LastRx Last sample
===============================================================================
#* GPS0                          0   4   377    11   -479ns[ -621ns] /-  134ns
^? a.b.c                         2   6   377    23   -923us[ -924us] +/-   43ms
^ d.e.f                         1   6   377    21  -2629us[-2619us] +/-   86ms
```

The columns are as follows:

**M**
- This indicates the mode of the source. ^ means a server, = means a peer and # indicates a locally connected reference clock.

**S**
- This column indicates the state of the sources. "*" indicates the source to which chronyd is currently synchronized. "+" indicates acceptable sources which are combined with the selected source. "-" indicates acceptable sources which are excluded by the combining algorithm. "?" indicates sources to which connectivity has been lost or whose packets do not pass all tests. "x" indicates a clock which chronyd thinks is a falseticker (its time is inconsistent with a majority of other sources). "~" indicates a source whose time appears to have too much variability. The "?" condition is also shown at start-up, until at least 3 samples have been gathered from it.

**Name/IP address**
- This shows the name or the IP address of the source, or reference ID for reference clock.

**Stratum**
- This shows the stratum of the source, as reported in its most recently received sample. Stratum 1 indicates a computer with a locally attached reference clock. A computer that is synchronized to a stratum 1 computer is at stratum 2. A computer that is synchronized to a stratum 2 computer is at stratum 3, and so on.

**Poll**
- This shows the rate at which the source is being polled, as a base-2 logarithm of the interval in seconds. Thus, a value of 6 would indicate that a measurement is being made every 64 seconds. chronyd automatically varies the polling rate in response to prevailing conditions.

**Reach**
- This shows the source’s reach register printed as an octal number. The register has 8 bits and is updated on every received or missed packet from the source. A value of 377 indicates that a valid reply was received for all of the last eight transmissions.

**LastRx**
- This column shows how long ago the last sample was received from the source. This is normally in seconds. The letters m, h, d or y indicate minutes, hours, days or years. A value of 10 years indicates there were no samples received from this source yet.

**Last sample**
- This column shows the offset between the local clock and the source at the last measurement. The number in the square brackets shows the actual measured offset. This may be suffixed by ns (indicating nanoseconds), us (indicating microseconds), ms (indicating milliseconds), or s (indicating seconds). The number to the left of the square brackets shows the original measurement, adjusted to allow for any slews applied to the local clock since. The number following the +/- indicator shows the margin of error in the measurement. Positive offsets indicate that the local clock is ahead of the source.
5.6.5.3. Checking chrony source statistics

The `sourcestats` command displays information about the drift rate and offset estimation process for each of the sources currently being examined by `chronyd`.

The optional argument `-v` can be specified, meaning verbose. In this case, extra caption lines are shown as a reminder of the meanings of the columns.

```bash
chronyc sourcestats
210 Number of sources = 1
Name/IP Address   NP   NR  Span  Frequency  Freq Skew  Offset  Std Dev
===============================================================================
abc.def.ghi        11   5   46m  -0.001      0.045      1us    25us
```

The columns are as follows:

**Name/IP address**

This is the name or IP address of the NTP server (or peer) or reference ID of the reference clock to which the rest of the line relates.

**NP**

This is the number of sample points currently being retained for the server. The drift rate and current offset are estimated by performing a linear regression through these points.

**NR**

This is the number of runs of residuals having the same sign following the last regression. If this number starts to become too small relative to the number of samples, it indicates that a straight line is no longer a good fit to the data. If the number of runs is too low, `chronyd` discards older samples and re-runs the regression until the number of runs becomes acceptable.

**Span**

This is the interval between the oldest and newest samples. If no unit is shown the value is in seconds. In the example, the interval is 46 minutes.

**Frequency**

This is the estimated residual frequency for the server, in parts per million. In this case, the computer’s clock is estimated to be running 1 part in $10^9$ slow relative to the server.

**Freq Skew**

This is the estimated error bounds on Freq (again in parts per million).

**Offset**

This is the estimated offset of the source.

**Std Dev**

This is the estimated sample standard deviation.

5.6.6. Manually Adjusting the System Clock

To step the system clock immediately, bypassing any adjustments in progress by slewing, issue the following command as `root`:

```
chronyc makestep
```

If the `rtcfile` directive is used, the real-time clock should not be manually adjusted. Random adjustments would interfere with chrony’s need to measure the rate at which the real-time clock drifts.
5.7. SETTING UP CHRONY FOR DIFFERENT ENVIRONMENTS

5.7.1. Setting up chrony for a system in an isolated network

For a network that is never connected to the Internet, one computer is selected to be the master timeserver. The other computers are either direct clients of the master, or clients of clients. On the master, the drift file must be manually set with the average rate of drift of the system clock. If the master is rebooted, it will obtain the time from surrounding systems and calculate an average to set its system clock. Thereafter it resumes applying adjustments based on the drift file. The drift file will be updated automatically when the `settime` command is used.

On the system selected to be the master, using a text editor running as root, edit `/etc/chrony.conf` as follows:

```
driftfile /var/lib/chrony/drift
commandkey 1
keyfile /etc/chrony.keys
initstepslew 10 client1 client3 client6
local stratum 8
manual
allow 192.0.2.0
```

Where **192.0.2.0** is the network or subnet address from which the clients are allowed to connect.

On the systems selected to be direct clients of the master, using a text editor running as root, edit the `/etc/chrony.conf` as follows:

```
server master
driftfile /var/lib/chrony/drift
logdir /var/log/chrony
log measurements statistics tracking
keyfile /etc/chrony.keys
commandkey 24
local stratum 10
initstepslew 20 master
allow 192.0.2.123
```

Where **192.0.2.123** is the address of the master, and **master** is the host name of the master. Clients with this configuration will resynchronize the master if it restarts.

On the client systems which are not to be direct clients of the master, the `/etc/chrony.conf` file should be the same except that the `local` and `allow` directives should be omitted.

In an isolated network, you can also use the `local` directive that enables a local reference mode, which allows `chronyd` operating as an NTP server to appear synchronized to real time, even when it was never synchronized or the last update of the clock happened a long time ago.

To allow multiple servers in the network to use the same local configuration and to be synchronized to one another, without confusing clients that poll more than one server, use the `orphan` option of the `local` directive which enables the orphan mode. Each server needs to be configured to poll all other servers with `local`. This ensures that only the server with the smallest reference ID has the local reference active and other servers are synchronized to it. When the server fails, another one will take over.
5.8. CHRONY WITH HW TIMESTAMPING

5.8.1. Understanding hardware timestamping

Hardware timestamping is a feature supported in some Network Interface Controller (NICs) which provides accurate timestamping of incoming and outgoing packets. NTP timestamps are usually created by the kernel and *chronyd* with the use of the system clock. However, when HW timestamping is enabled, the NIC uses its own clock to generate the timestamps when packets are entering or leaving the link layer or the physical layer. When used with NTP, hardware timestamping can significantly improve the accuracy of synchronization. For best accuracy, both NTP servers and NTP clients need to use hardware timestamping. Under ideal conditions, a sub-microsecond accuracy may be possible.

Another protocol for time synchronization that uses hardware timestamping is PTP.

Unlike NTP, PTP relies on assistance in network switches and routers. If you want to reach the best accuracy of synchronization, use PTP on networks that have switches and routers with PTP support, and prefer NTP on networks that do not have such switches and routers.

5.8.2. Verifying support for hardware timestamping

To verify that hardware timestamping with NTP is supported by an interface, use the `ethtool -T` command. An interface can be used for hardware timestamping with NTP if `ethtool` lists the *SOF_TIMESTAMPING_TX_HARDWARE* and *SOF_TIMESTAMPING_TX_SOFTWARE* capabilities and also the *HWTSTAMP_FILTER_ALL* filter mode.

**Example 5.1. Verifying support for hardware timestamping on a specific interface**

```bash
~# ethtool -T eth0
```

Output:

```
Timestamping parameters for eth0:
Capabilities:
  hardware-transmit     (SOF_TIMESTAMPING_TX_HARDWARE)
  software-transmit     (SOF_TIMESTAMPING_TX_SOFTWARE)
  hardware-receive      (SOF_TIMESTAMPING_RX_HARDWARE)
  software-receive      (SOF_TIMESTAMPING_RX_SOFTWARE)
  software-system-clock (SOF_TIMESTAMPING_SOFTWARE)
  hardware-raw-clock    (SOF_TIMESTAMPING_RAW_HARDWARE)
PTP Hardware Clock: 0
Hardware Transmit Timestamp Modes:
  off                   (HWTSTAMP_TX_OFF)
  on                    (HWTSTAMP_TX_ON)
Hardware Receive Filter Modes:
  none                  (HWTSTAMP_FILTER_NONE)
  all                   (HWTSTAMP_FILTER_ALL)
  ptpv1-l4-sync         (HWTSTAMP_FILTER_PTP_V1_L4_SYNC)
  ptpv1-l4-delay-req    (HWTSTAMP_FILTER_PTP_V1_L4_DELAY_REQ)
  ptpv2-l4-sync         (HWTSTAMP_FILTER_PTP_V2_L4_SYNC)
  ptpv2-l4-delay-req    (HWTSTAMP_FILTER_PTP_V2_L4_DELAY_REQ)
  ptpv2-l2-sync         (HWTSTAMP_FILTER_PTP_V2_L2_SYNC)
  ptpv2-l2-delay-req    (HWTSTAMP_FILTER_PTP_V2_L2_DELAY_REQ)
```

5.8.3. Enabling hardware timestamping

To enable hardware timestamping, use the `hwtimestamp` directive in the `/etc/chrony.conf` file. The directive can either specify a single interface, or a wildcard character can be used to enable hardware timestamping on all interfaces that support it. Use the wildcard specification in case that no other application, like `ptp4l` from the `linuxptp` package, is using hardware timestamping on an interface. Multiple `hwtimestamp` directives are allowed in the chrony configuration file.

Example 5.2. Enabling hardware timestamping by using the `hwtimestamp` directive

```
hwtimestamp eth0
hwtimestamp eth1
hwtimestamp *
```

5.8.4. Configuring client polling interval

The default range of a polling interval (64-1024 seconds) is recommended for servers on the Internet. For local servers and hardware timestamping, a shorter polling interval needs to be configured in order to minimize offset of the system clock.

The following directive in `/etc/chrony.conf` specifies a local NTP server using one second polling interval:

```
server ntp.local minpoll 0 maxpoll 0
```

5.8.5. Enabling interleaved mode

NTP servers that are not hardware NTP appliances, but rather general purpose computers running a software NTP implementation, like `chrony`, will get a hardware transmit timestamp only after sending a packet. This behavior prevents the server from saving the timestamp in the packet to which it corresponds. In order to enable NTP clients receiving transmit timestamps that were generated after the transmission, configure the clients to use the NTP interleaved mode by adding the `xleave` option to the server directive in `/etc/chrony.conf`:

```
server ntp.local minpoll 0 maxpoll 0 xleave
```

5.8.6. Configuring server for large number of clients

The default server configuration allows a few thousands of clients at most to use the interleaved mode concurrently. To configure the server for a larger number of clients, increase the `clientloglimit` directive in `/etc/chrony.conf`. This directive specifies the maximum size of memory allocated for logging of clients’ access on the server:

```
clientloglimit 100000000
```
5.8.7. Verifying hardware timestamping

To verify that the interface has successfully enabled hardware timestamping, check the system log. The log should contain a message from `chronyd` for each interface with successfully enabled hardware timestamping.

Example 5.3. Log messages for interfaces with enabled hardware timestamping

```
chronyd[4081]: Enabled HW timestamping on eth0
chronyd[4081]: Enabled HW timestamping on eth1
```

When `chronyd` is configured as an NTP client or peer, you can have the transmit and receive timestamping modes and the interleaved mode reported for each NTP source by the `chronyc ntpdata` command:

Example 5.4. Reporting the transmit, receive timestamping and interleaved mode for each NTP source

```
~]# chronyc ntpdata
```

```
Remote address : 203.0.113.15 (CB00710F)
Remote port     : 123
Local address   : 203.0.113.74 (CB00714A)
Leap status     : Normal
Version         : 4
Mode            : Server
Stratum         : 1
Poll interval   : 0 (1 seconds)
Precision       : -24 (0.000000060 seconds)
Root delay      : 0.000015 seconds
Root dispersion : 0.000015 seconds
Reference ID    : 47505300 (GPS)
Reference time  : Wed May 03 13:47:45 2017
Offset          : -0.000000134 seconds
Peer delay      : 0.000005396 seconds
Peer dispersion : 0.000002329 seconds
Response time   : 0.000152073 seconds
Jitter asymmetry: +0.00
NTP tests       : 111 111 1111
Interleaved     : Yes
Authenticated   : No
TX timestamping : Hardware
RX timestamping : Hardware
Total TX        : 27
Total RX        : 27
Total valid RX  : 27
```

Example 5.5. Reporting the stability of NTP measurements
With hardware timestamping enabled, stability of NTP measurements should be in tens or hundreds of nanoseconds, under normal load. This stability is reported in the Std Dev column of the output of the `chronyc sourcestats` command:

```
Output:
210 Number of sources = 1
Name/IP Address           NP  NR  Span  Frequency  Freq Skew  Offset  Std Dev
ntp.local                12   7   11   +0.000      0.019     +0ns    49ns
```

5.8.8. Configuring PTP-NTP bridge

If a highly accurate Precision Time Protocol (PTP) grandmaster is available in a network that does not have switches or routers with PTP support, a computer may be dedicated to operate as a PTP slave and a stratum-1 NTP server. Such a computer needs to have two or more network interfaces, and be close to the grandmaster or have a direct connection to it. This will ensure highly accurate synchronization in the network.

Configure the `ptp4l` and `phc2sys` programs from the `linuxptp` packages to use one interface to synchronize the system clock using PTP.

Configure `chronyd` to provide the system time using the other interface:

```
Example 5.6. Configuring chronyd to provide the system time using the other interface

bindaddress 203.0.113.74
hwtimestamp eth1
local stratum 1
```

5.9. ACHIEVING SOME SETTINGS PREVIOUSLY SUPPORTED BY NTP IN CHRONY

Some settings that were in previous major version of Red Hat Enterprise Linux supported by ntp, are not supported by chrony. This section lists such settings, and describes ways to achieve them on a system with chrony.

5.9.1. Monitoring by ntpq and ntpdc

`chronyd` cannot be monitored by the `ntpq` and `ntpd` utilities from the `ntp` distribution, because `chrony` does not support the NTP modes 6 and 7. It supports a different protocol and `chronyc` is the client implementation. For more information, see the `chronyc(1)` man page.

To monitor the status of the system clock synchronized by `chronyd`, you can:

- Use the tracking command
- Use the `ntpstat` utility, which supports `chrony` and provides a similar output as it used to with `ntpd`
Example 5.7. Using the tracking command

```bash
$ chronyc -n tracking
Reference ID    : 0A051B0A (10.5.27.10)
Stratum         : 2
Ref time (UTC)  : Thu Mar 08 15:46:20 2018
System time     : 0.000000338 seconds slow of NTP time
Last offset     : +0.000339408 seconds
RMS offset      : 0.000339408 seconds
Frequency       : 2.968 ppm slow
Residual freq   : +0.001 ppm
Skew            : 3.336 ppm
Root delay      : 0.157559142 seconds
Root dispersion : 0.001339232 seconds
Update interval : 64.5 seconds
Leap status     : Normal
```

Example 5.8. Using the ntpstat utility

```bash
$ ntpstat
synchronised to NTP server (10.5.27.10) at stratum 2
time correct to within 80 ms
polling server every 64 s
```

5.9.2. Using authentication mechanism based on public key cryptography

In Red Hat Enterprise Linux 7, `ntp` supported `Autokey`, which is an authentication mechanism based on public key cryptography. `Autokey` is not supported in `chronyd`.

On a Red Hat Enterprise Linux 8 system, it is recommended to use symmetric keys instead. Generate the keys with the `chronyc keygen` command. A client and server need to share a key specified in `/etc/chrony.keys`. The client can enable authentication using the `key` option in the `server`, `pool`, or `peer` directive.

5.9.3. Using ephemeral symmetric associations

In Red Hat Enterprise Linux 7, `ntpd` supported ephemeral symmetric associations, which can be mobilized by packets from peers which are not specified in the `ntp.conf` configuration file. In Red Hat Enterprise Linux 8, `chronyd` needs all peers to be specified in `chrony.conf`. Ephemeral symmetric associations are not supported.

Note that using the client/server mode enabled by the `server` or `pool` directive is more secure compared to the symmetric mode enabled by the `peer` directive.

5.9.4. multicast/broadcast client

Red Hat Enterprise Linux 7 supported the broadcast/multicast NTP mode, which simplifies configuration of clients. With this mode, clients can be configured to just listen for packets sent to a multicast/broadcast address instead of listening for specific names or addresses of individual servers, which may change over time.
In Red Hat Enterprise Linux 8, **chronyd** does not support the broadcast/multicast mode. The main reason is that it is less accurate and less secure than the ordinary client/server and symmetric modes.

There are several options of migration from an **NTP** broadcast/multicast setup:

- Configure DNS to translate a single name, such as ntp.example.com, to multiple addresses of different servers
  Clients can have a static configuration using only a single pool directive to synchronize with multiple servers. If a server from the pool becomes unreachable, or otherwise unsuitable for synchronization, the clients automatically replace it with another server from the pool.

- Distribute the list of **NTP** servers over DHCP
  When NetworkManager gets a list of **NTP** servers from the DHCP server, **chronyd** is automatically configured to use them. This feature can be disabled by adding `PEERNTP=no` to the `/etc/sysconfig/network` file.

- Use the **Precision Time Protocol (PTP)**
  This option is suitable mainly for environments where servers change frequently, or if a larger group of clients needs to be able to synchronize to each other without having a designated server.

  **PTP** was designed for multicast messaging and works similarly to the **NTP** broadcast mode. A **PTP** implementation is available in the **linuxptp** package.

  **PTP** normally requires hardware timestamping and support in network switches to perform well. However, **PTP** is expected to work better than **NTP** in the broadcast mode even with software timestamping and no support in network switches.

  In networks with very large number of **PTP** slaves in one communication path, it is recommended to configure the **PTP** slaves with the `hybrid_e2e` option in order to reduce the amount of network traffic generated by the slaves. You can configure a computer running **chronyd** as an **NTP** client, and possibly **NTP** server, to operate also as a **PTP** grandmaster to distribute synchronized time to a large number of computers using multicast messaging.

### 5.10. ADDITIONAL RESOURCES

The following sources of information provide additional resources regarding **chrony**.

#### 5.10.1. Installed Documentation

- **chronyc(1)** man page – Describes the **chronyc** command-line interface tool including commands and command options.

- **chronyd(8)** man page – Describes the **chronyd** daemon including commands and command options.

- **chrony.conf(5)** man page – Describes the **chrony** configuration file.

#### 5.10.2. Online Documentation


For answers to FAQs, see https://chrony.tuxfamily.org/faq.html

5.11. MANAGING TIME SYNCHRONIZATION USING RHEL SYSTEM ROLES

You can manage time synchronization on multiple target machines using the `timesync` role.

The `timesync` role installs and configures an NTP or PTP implementation to operate as an NTP client or PTP slave in order to synchronize the system clock with NTP servers or grandmasters in PTP domains.

Note that using the `timesync` role also facilitates migration to `chrony`, because you can use the same playbook on all versions of Red Hat Enterprise Linux starting with RHEL 6 regardless of whether the system uses `ntp` or `chrony` to implement the NTP protocol.

**WARNING**

The `timesync` role replaces the configuration of the given or detected provider service on the managed host. Previous settings are lost, even if they are not specified in the role variables. The only preserved setting is the choice of provider if the `timesync_ntp_provider` variable is not defined.

The following example shows how to apply the `timesync` role in a situation with just one pool of servers.

```
Example 5.9. An example playbook applying the timesync role for a single pool of servers

---
- hosts: timesync-test
  vars:
    timesync_ntp_servers:
      - hostname: 2.rhel.pool.ntp.org
        pool: yes
        iburst: yes
    roles:
      - rhel-system-roles.timesync
```

For more information on using the `timesync` role, see System roles documentation.
6.1. INTRODUCTION TO PYTHON

Python is a high-level programming language that supports multiple programming paradigms, such as object-oriented, imperative, functional, and procedural. Python has dynamic semantics and can be used for general-purpose programming.

With Red Hat Enterprise Linux, many packages that are installed on the system, such as packages providing system tools, tools for data analysis or web applications are written in Python. To be able to use these packages, you need to have the python packages installed.

6.1.1. Python versions

Two incompatible versions of Python are widely used, Python 2.x and Python 3.x.

Red Hat Enterprise Linux 8 uses Python 3.6 by default. However, Python 2.7 is also provided to support existing software.

WARNING

Neither the default python package nor the unversioned /usr/bin/python executable is distributed with Red Hat Enterprise Linux 8.

IMPORTANT

Always specify the major version of Python when installing it, invoking it, or otherwise interacting with it. For example, use python3, instead of python, in package and command names. All Python-related commands should also include the version, for example, pip3 or pip2.

Alternatively, configure the system default version by using the alternatives command as described in Configuring the unversioned Python.

As a system administrator, you are recommended to use preferably Python 3 for the following reasons:

- Python 3 represents the main development direction of the Python project.
- Support for Python 2 in the upstream community ends in 2020.
- Popular Python libraries are dropping Python 2 support in upstream.
- Python 2 in Red Hat Enterprise Linux 8 will have a shorter life cycle and its aim is to facilitate smoother transition to Python 3 for customers.

For developers, Python 3 has the following advantages over Python 2:

- Python 3 allows writing expressive, maintainable, and correct code more easily.
• Code written in Python 3 will have greater longevity.

• Python 3 has new features, including asyncio, f-strings, advanced unpacking, keyword only arguments, chained exceptions and more.

However, existing software tends to require \texttt{/usr/bin/python} to be Python 2. For this reason, no default \texttt{python} package is distributed with Red Hat Enterprise Linux 8, and you can choose between using Python 2 and 3 as \texttt{/usr/bin/python}, as described in Section 6.2.3, “Configuring the unversioned Python”.

### 6.1.2. The internal platform-python package

System tools in Red Hat Enterprise Linux 8 use a Python version 3.6 provided by the internal \texttt{platform-python} package. Red Hat advises customers to use the \texttt{python36} package instead.

### 6.2. INSTALLING AND USING PYTHON

#### WARNING

Using the unversioned \texttt{python} command to install or run Python does not work by default due to ambiguity. Always specify the major version of Python as described in Installing and using Python 3 and Installing and using Python 2, or configure the system default version by using the \texttt{alternatives} command as described in Section 6.2.3, “Configuring the unversioned Python”.

#### 6.2.1. Installing and using Python 3

In Red Hat Enterprise Linux 8, Python 3 is distributed as the \texttt{python36} module in the AppStream repository.

For details regarding modules, see Installing, managing, and removing user space components.

##### 6.2.1.1. Installing Python 3

To install Python 3, use the following command as root:

```
yum install python3
```

This command installs Python 3.6 from the \texttt{python36} module in AppStream.

##### 6.2.1.2. Using Python 3

To run Python 3, use the \texttt{python3} command. Use the version in all other related commands, such as \texttt{pip3}.

##### 6.2.1.3. Naming conventions for Python 3 packages

Packages with add-on modules for Python 3 generally use the \texttt{python3-} prefix.

For example, to install the Requests module that is used for writing HTTP clients, run:
yum install python3-requests

6.2.2. Installing and using Python 2

Some software has not yet been fully ported to Python 3, and needs Python 2 to operate. Red Hat Enterprise Linux 8 allows parallel installation of Python 3 and Python 2. If you need the Python 2 functionality, install the `python27` module, which is available in the AppStream repository.

For details regarding modules, see Installing, managing, and removing user space components.

WARNING
Note that Python 3 is the main development direction of the Python project. The support for Python 2 is being phased out. The `python27` module has a shorter support period than Red Hat Enterprise Linux 8.

6.2.2.1. Installing Python 2

To install Python 2, run as root:

```
yum install python2
```

This command installs Python 2.7 from the `python27` module in AppStream.

6.2.2.2. Using Python 2

To run Python 2, use the `python2` command. Use the version in all other related commands, such as `pip2`.

6.2.2.2.1. Naming conventions for Python 2 packages

Packages with add-on modules for Python 2 generally use the `python2-` prefix.

For example, to install the Requests module that is used for writing HTTP clients, run:

```
yum install python2-requests
```

6.2.3. Configuring the unversioned Python

System administrators can configure the unversioned `python` command on the system using the `alternatives` command. Note that the required package, either `python3` or `python2`, needs to be installed before configuring the unversioned command to the respective version.

To configure the unversioned `python` command to Python 3 directly, run:

```
alternatives --set python /usr/bin/python3
```

Use an analogous command if you choose Python 2.
Alternatively, you can configure the unversioned `python` command interactively:

1. Run the following command:
   ```bash
   alternatives --config python
   ```
2. Select the required version from the provided list.

To reset this configuration and remove the unversioned `python` command, run:

```bash
alternatives --auto python
```

**WARNING**

Additional Python-related commands, such as `pip3`, do not have configurable unversioned variants.

---

### 6.3. MIGRATING FROM PYTHON 2 TO PYTHON 3

As a developer, you may want to migrate your former code that is written in Python 2 to Python 3. For more information on how to migrate large code bases to Python 3, see [The Conservative Python 3 Porting Guide](#).

Note that after this migration, the original Python 2 code becomes interpretable by the Python 3 interpreter and stays interpretable for the Python 2 interpreter as well.

### 6.4. PACKAGING OF PYTHON 3 RPMS

Most Python projects use Setuptools for packaging, and define package information in the `setup.py` file. For more information on Setuptools packaging, see [Setuptools documentation](#).

You can also package your Python project into an RPM package, which provides the following advantages compared to Setuptools packaging:

- Specification of dependencies of a package on other RPMs (even non-Python)
- Cryptographic signing
  With cryptographic signing, content of RPM packages can be verified, integrated, and tested with the rest of the operating system.

For more information on RPM packaging, see [Packaging and distributing software](#).

#### 6.4.1. Typical SPEC file description for a Python RPM package

A SPEC file contains instructions that the `rpmbuild` utility uses to build an RPM. The instructions are included in a series of sections. A SPEC file has two main parts in which the sections are defined:

- Preamble (contains a series of metadata items that are used in the Body)
Body (contains the main part of the instructions)

For further information about SPEC files, see Packaging and distributing software.

An RPM SPEC file for Python projects has some specifics compared to non-Python RPM SPEC files. Most notably, a name of any RPM package of a Python library must always include the `python3` prefix.

Other specifics are shown in the following SPEC file example for the `python3-detox` package. For description of such specifics, see the notes below the example.

```spec
%global modname detox

Name:           python3-detox
Version:        0.12
Release:        4%{?dist}
Summary:        Distributing activities of the tox tool
License:        MIT
URL:            https://pypi.io/project/detox
Source0:        https://pypi.io/packages/source/d/%{modname}/%{modname}-%{version}.tar.gz
BuildArch:      noarch

BuildRequires:  python36-devel
BuildRequires:  python3-setuptools
BuildRequires:  python36-rpm-macros
BuildRequires:  python3-six
BuildRequires:  python3-tox
BuildRequires:  python3-py
BuildRequires:  python3-eventlet

%?python_enable_dependency_generator

%description

Detox is the distributed version of the tox python testing tool. It makes efficient use of multiple CPUs by running all possible activities in parallel. Detox has the same options and configuration that tox has, so after installation you can run it in the same way and with the same options that you use for tox.

$ detox

%prep
%autosetup -n %{modname}-%{version}

%build
%py3_build

%install
%py3_install

%check
%{__python3} setup.py test

%files -n python3-%{modname}
%doc CHANGESLOG
```
The \texttt{modname} macro contains the name of the Python project. In this example it is \texttt{detox}.

When packaging a Python project into RPM, the \texttt{python3} prefix always needs to be added to the original name of the project. The original name here is \texttt{detox} and the name of the RPM is \texttt{python3-detox}.

\textbf{BuildRequires} specifies what packages are required to build and test this package. In BuildRequires, always include items providing tools necessary for building Python packages: \texttt{python36-devel} and \texttt{python3-setuptools}. The \texttt{python36-rpm-macros} package is required so that files with \texttt{/usr/bin/python3} shebangs are automatically changed to \texttt{/usr/bin/python3.6}. For more information, see Section 6.4.4, “Handling hashbangs in Python scripts”.

Every Python package requires some other packages to work correctly. Such packages need to be specified in the SPEC file as well. To specify the dependencies, you can use the \texttt{%python_enable_dependency_generator} macro to automatically use dependencies defined in the \texttt{setup.py} file. If a package has dependencies that are not specified using Setuptools, specify them within additional \texttt{Requires} directives.

The \texttt{%py3\_build} and \texttt{%py3\_install} macros run the \texttt{setup.py build} and \texttt{setup.py install} commands, respectively, with additional arguments to specify installation locations, the interpreter to use, and other details.

The \texttt{check} section provides a macro that runs the correct version of Python. The \texttt{%{__python3}} macro contains a path for the Python 3 interpreter, for example \texttt{/usr/bin/python3}. We recommend to always use the macro rather than a literal path.

### 6.4.2. Common macros for Python 3 RPM packages

In a SPEC file, always use the macros below rather than hardcoding their values.

In macro names, always use \texttt{python3} or \texttt{python2} instead of unversioned \texttt{python}.

<table>
<thead>
<tr>
<th>Macro</th>
<th>Normal Definition</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>%{__python3}</td>
<td>/usr/bin/python3</td>
<td>Python 3 interpreter</td>
</tr>
<tr>
<td>%{python3_version}</td>
<td>3.6</td>
<td>The full version of the Python 3 interpreter.</td>
</tr>
<tr>
<td>%{python3_sitelib}</td>
<td>/usr/lib/python3.6/site-packages</td>
<td>Where pure-Python modules are installed.</td>
</tr>
<tr>
<td>%{python3_sitearch}</td>
<td>/usr/lib64/python3.6/site-packages</td>
<td>Where modules containing architecture-specific extensions are installed.</td>
</tr>
<tr>
<td>Macro</td>
<td>Normal Definition</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>----------------------------------------------------------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>%py3_build</td>
<td>Runs the <code>setup.py build</code> command with arguments suitable for a system package.</td>
<td></td>
</tr>
<tr>
<td>%py3_install</td>
<td>Runs the <code>setup.py install</code> command with arguments suitable for a system package.</td>
<td></td>
</tr>
</tbody>
</table>

### 6.4.3. Automatic provides for Python RPM packages

When packaging a Python project, make sure that, if present, the following directories are included in the resulting RPM:

- `.dist-info`
- `.egg-info`
- `.egg-link`

From these directories, the RPM build process automatically generates virtual `pythonX.Ydist` provides, for example `python3.6dist(detox)`. These virtual provides are used by packages that are specified by the `%python_enable_dependency_generator` macro.

### 6.4.4. Handling hashbangs in Python scripts

In Red Hat Enterprise Linux 8, executable Python scripts are expected to use hashbangs (shebangs) specifying explicitly at least the major Python version.

The `/usr/lib/rpm/redhat/brp-mangle-shebangs` buildroot policy (BRP) script is run automatically when building any RPM package, and attempts to correct hashbangs in all executable files. The BRP script will generate errors when encountering a Python script with an ambiguous hashbang, such as:

```bash
#!/usr/bin/python
```

or

```bash
#!/usr/bin/env python
```

To modify hashbangs in the Python scripts causing these build errors at RPM build time, use the `pathfix.py` script from the `platform-python-devel` package:

```bash
pathfix.py -pn -i %{__python3} PATH ...
```
Multiple **PATHs** can be specified. If a **PATH** is a directory, **pathfix.py** recursively scans for any Python scripts matching the pattern `^[a-zA-Z0-9_]\+.py$`, not only those with an ambiguous hashbang. Add this command to the `%prep` section or at the end of the `%install` section.

Alternatively, modify the packaged Python scripts so that they conform to the expected format. For this purpose, **pathfix.py** can be used outside the RPM build process, too. When running **pathfix.py** outside a RPM build, replace `__python3` from the example above with a path for the hashbang, such as `/usr/bin/python3`.

If the packaged Python scripts require Python version 2, replace the number 3 with 2 in the commands above.

Additionally, hashbangs in the form `/usr/bin/python3` are by default replaced with hashbangs pointing to Python from the **platform-python** package used for system tools with Red Hat Enterprise Linux.

To change the `/usr/bin/python3` hashbangs in their custom packages to point to a version of Python installed from Application Stream, in the form `/usr/bin/python3.6`, add the **python36-rpm-macros** package into the **BuildRequires** section of the SPEC file:

```
BuildRequires: python36-rpm-macros
```

**NOTE**

To prevent hashbang check and modification by the BRP script, use the following RPM directive:

```
%define %brp_mangle_shebangs
```

```
CHAPTER 7. INSTALLING AND USING LANGPACKS

7.1. INTRODUCTION TO LANGPACKS

Langpacks are meta-packages which install extra add-on packages containing translations, dictionaries and locales for every package installed on the system.

On a Red Hat Enterprise Linux 8 system, langpacks installation is based on the langpacks-<langcode> language meta-packages and RPM weak dependencies (Supplements tag).

There are two prerequisites to be able to use langpacks for a selected language:

1. The langpacks-<langcode> language meta-package for the selected language has been installed on the system.
   On Red Hat Enterprise Linux 8, the langpacks meta packages are installed automatically with the initial installation of the operating system using the Anaconda installer, because these packages are available in the in Application Stream repository.

   To find which languages provide langpacks, execute the following command:

   ~]# yum list langpacks-*

2. The base package, for which you want to search the local packages, has already been installed on the system.

   If these prerequisites are fulfilled, the language meta-packages pull their langpack for the selected language automatically in the transaction set.

7.2. WORKING WITH RPM WEAK DEPENDENCY-BASED LANGPACKS

7.2.1. Querying RPM weak dependency-based langpacks

To list the already installed language support, run the following command:

~]# yum list installed langpacks*  

To check if any language support is available for another language, run the following command:

~]# yum list available langpacks*  

To list what packages get installed for any language, run the following command:

~]# yum repoquery --whatsonly=langpacks-<locale_code>

7.2.2. Installing language support

To add new a language support, run the following command as the root user:

~]# yum install langpacks-<locale_code>
7.2.3. Removing language support

To remove any installed language support, run the following command as the root user:

```
~# yum remove langpacks-<locale_code>
```

7.3. SAVING DISK SPACE BY USING GLIBC-LANGPACK-<LOCALE_CODE>

Currently, all locales are stored in the `/usr/lib/locale/locale-archive` file, which requires a lot of disk space.

On systems where disk space is a critical issue, such as containers and cloud images, or only a few locales are needed, you can use the glibc locale langpack packages (`glibc-langpack-<locale_code>`). By using the following command instead of that described in Section 7.2.2, “Installing language support”, you can install locales individually and thus gain a smaller package installation footprint:

```
~# yum install glibc-langpack-<locale_code>
```

When installing the operating system with Anaconda, `glibc-langpack-<locale_code>` is installed for the language you used during the installation and also for the languages you selected as additional languages. Note that `glibc-all-langpacks`, which contains all locales, is installed by default, so some locales are duplicated. If you installed `glibc-langpack-<locale_code>` for one or more selected languages, you can delete `glibc-all-langpacks` after the installation to save the disk space.

Note that installing only selected `glibc-langpack-<locale_code>` packages instead of `glibc-all-langpacks` has impact on run time performance.

**NOTE**

If disk space is not an issue, keep all locales installed by using the `glibc-all-langpacks` package.
CHAPTER 8. GETTING STARTED WITH TCL/TK ON RED HAT ENTERPRISE LINUX 8

8.1. INTRODUCTION TO TCL/TK

Tool command language (Tcl) is a dynamic programming language. The interpreter for this language, together with the C library, is provided by the tcl package.

Using Tcl paired with Tk (Tcl/Tk) enables creating cross-platform GUI applications. Tk is provided by the tk package.

Note that Tk can refer to any of the the following:

- A programming toolkit for multiple languages
- A Tk C library bindings available for multiple languages, such as C, Ruby, Perl and Python
- A wish interpreter that instantiates a Tk console
- A Tk extension that adds a number of new commands to a particular Tcl interpreter

For more information about Tcl/Tk, see the Tcl/Tk manual or Tcl/Tk documentation web page.

8.2. NOTABLE CHANGES IN TCL/TK 8.6

Major changes in Tcl/Tk 8.6 compared to Tcl/Tk 8.5 are:

- Object-oriented programming support
- Stackless evaluation implementation
- Enhanced exceptions handling
- Collection of third-party packages built and installed with Tcl
- Multi-thread operations enabled
- SQL database-powered scripts support
- IPv6 networking support
- Built-in Zlib compression
- List processing
  Two new commands, lmap and dict map are available, which allow the expression of transformations over Tcl containers.
- Stacked channels by script
  Two new commands, chan push and chan pop are available, which allow to add or remove transformations to or from I/O channels.

Major changes in Tk include:

- Built-in PNG image support
• Busy windows
  A new command, **tk busy** is available, which disables user interaction for a window or a widget and shows the busy cursor.

• New font selection dialog interface

• Angled text support

• Moving things on a canvas support

For the detailed list of changes between **Tcl 8.5** and **Tcl 8.6**, see [Changes in Tcl/Tk 8.6](#).

### 8.3. MIGRATING TO TCL/TK 8.6

Red Hat Enterprise Linux 7 used **Tcl/Tk 8.5**. With Red Hat Enterprise Linux 8, **Tcl/Tk version 8.6** is provided in the Base OS repository.

This section describes migration path to **Tcl/Tk 8.6** for:

• developers writing **Tcl** extensions or embedding **Tcl** interpreter into their applications (see Section 8.3.1, "Migration path for developers of Tcl extensions")

• users scripting tasks with **Tcl/Tk** (see Section 8.3.2, "Migration path for users scripting their tasks with Tcl/Tk")

#### 8.3.1. Migration path for developers of Tcl extensions

**Tcl 8.6** limits direct access to members of the **interp** structure. For example, if your code reads `interp -> errorLine`, rewrite the code to use the **Tcl_GetErrorLine(interp)** function.

To make your code compatible with both **Tcl 8.5** and **Tcl 8.6**, use the following code snippet in a header file of your C or C++ application or extension that includes the **Tcl** library:

```c
#include <tcl.h>
#if !defined(Tcl_GetErrorLine)
#define Tcl_GetErrorLine(interp) (interp -> errorLine)
#endif
```

#### 8.3.2. Migration path for users scripting their tasks with Tcl/Tk

In **Tcl 8.6**, most scripts work the same way as with the previous version of **Tcl**. When writing a portable code, make sure to not use the commands that are no longer supported in **Tk 8.6**:  

• `tkIconList_Arrange`

• `tkIconList_AutoScan`

• `tkIconList_Btn1`

• `tkIconList_Config`

• `tkIconList_Create`

• `tkIconList_CtrlBtn1`
You can check the list of unsupported commands also in the /usr/share/tk8.6/unsupported.tcl file.
CHAPTER 9. USING PREFIXDEVNAME FOR NAMING OF ETHERNET NETWORK INTERFACES

This documentation describes how to set the prefixes for consistent naming of Ethernet network interfaces in case that you do not want to use the default naming scheme of such interfaces.

However, Red Hat recommends to use the default naming scheme, which is the same as in Red Hat Enterprise Linux 7. For more details about this scheme, see Consistent Network Device Naming.

9.1. INTRODUCTION TO PREFIXDEVNAME

The `prefixdevname` tool is a udev helper utility that enables you to define your own prefix used for naming of the Ethernet network interfaces.

9.2. SETTING PREFIXDEVNAME

The setting of the prefix with `prefixdevname` is done during system installation.

To set and activate the required prefix for your Ethernet network interfaces, add the following on the kernel command line:

```
net.ifnames.prefix=<required prefix>
```

**WARNING**

Red Hat does not support the use of `prefixdevname` on already deployed systems.

After the prefix was once set, and the operating system was rebooted, the prefix is effective every time when a new network interface appears. The new device is assigned a name in the form of `<PREFIX> <INDEX>`. For example, if your selected prefix is `net`, and the interfaces with `net0` and `net1` prefixes already exist on the system, the new interface is named `net2`. The `prefixdevname` utility then generates the new `.link` file in the `/etc/systemd/network` directory that applies the name to the interface with the MAC address that just appeared. The configuration is persistent across reboots.

9.3. LIMITATIONS OF PREFIXDEVNAME

There are certain limitations for prefixes of Ethernet network interfaces.

The prefix that you choose must meet the following requirements:

- be ASCII string
- be alphanumeric string
- be shorter than 16 characters
WARNING

The prefix cannot conflict with any other well-known prefix used for network interface naming on Linux. Specifically, you cannot use these prefixes: eth, eno, ens, em.