



Red Hat Enterprise Linux 8

Configuring basic system settings

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

Red Hat Enterprise Linux 8 Configuring basic system settings

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

Legal Notice

Copyright © 2021 Red Hat, Inc.

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

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

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

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

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

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

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

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

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

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

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

All other trademarks are the property of their respective owners.

Abstract

This 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

MAKING OPEN SOURCE MORE INCLUSIVE	9
PROVIDING FEEDBACK ON RED HAT DOCUMENTATION	10
CHAPTER 1. GETTING STARTED WITH RHEL SYSTEM ROLES	11
1.1. INTRODUCTION TO RHEL SYSTEM ROLES	11
1.2. RHEL SYSTEM ROLES TERMINOLOGY	11
1.3. APPLYING A ROLE	12
1.4. ADDITIONAL RESOURCES	14
CHAPTER 2. CHANGING BASIC ENVIRONMENT SETTINGS	15
2.1. CONFIGURING THE DATE AND TIME	15
2.1.1. Displaying the current date and time	15
2.2. CONFIGURING THE SYSTEM LOCALE	15
2.3. CONFIGURING THE KEYBOARD LAYOUT	16
2.4. CHANGING THE LANGUAGE USING DESKTOP GUI	17
2.5. ADDITIONAL RESOURCES	19
CHAPTER 3. CONFIGURING AND MANAGING NETWORK ACCESS	20
3.1. CONFIGURING THE NETWORK AND HOST NAME IN THE GRAPHICAL INSTALLATION MODE	20
3.2. CONFIGURING A STATIC ETHERNET CONNECTION USING NMCLI	21
3.3. ADDING A CONNECTION PROFILE USING NMTUI	24
3.4. MANAGING NETWORKING IN THE RHEL WEB CONSOLE	26
3.5. MANAGING NETWORKING USING RHEL SYSTEM ROLES	27
3.6. ADDITIONAL RESOURCES	28
CHAPTER 4. REGISTERING THE SYSTEM AND MANAGING SUBSCRIPTIONS	29
4.1. REGISTERING THE SYSTEM AFTER THE INSTALLATION	29
4.2. REGISTERING SUBSCRIPTIONS WITH CREDENTIALS IN THE WEB CONSOLE	30
4.3. REGISTERING A SYSTEM USING RED HAT ACCOUNT ON GNOME	33
4.4. REGISTERING A SYSTEM USING AN ACTIVATION KEY ON GNOME	34
4.5. REGISTERING RHEL 8.4 USING THE INSTALLER GUI	34
CHAPTER 5. MAKING SYSTEMD SERVICES START AT BOOT TIME	36
5.1. ENABLING OR DISABLING THE SERVICES	36
5.2. MANAGING SERVICES IN THE RHEL WEB CONSOLE	36
CHAPTER 6. CONFIGURING SYSTEM SECURITY	39
6.1. ENABLING THE FIREWALLD SERVICE	39
6.2. MANAGING FIREWALL IN THE RHEL 8 WEB CONSOLE	40
6.3. MANAGING BASIC SELINUX SETTINGS	40
6.4. ENSURING THE REQUIRED STATE OF SELINUX	41
6.5. SWITCHING SELINUX MODES IN THE RHEL 8 WEB CONSOLE	42
6.6. ADDITIONAL RESOURCES	42
CHAPTER 7. GETTING STARTED WITH MANAGING USER ACCOUNTS	44
7.1. MANAGING ACCOUNTS AND GROUPS USING COMMAND LINE TOOLS	44
7.2. SYSTEM USER ACCOUNTS MANAGED IN THE WEB CONSOLE	45
7.3. ADDING NEW ACCOUNTS USING THE WEB CONSOLE	45
CHAPTER 8. DUMPING A CRASHED KERNEL FOR LATER ANALYSIS	47
8.1. WHAT IS KDUMP	47
8.2. CONFIGURING KDUMP MEMORY USAGE AND TARGET LOCATION IN WEB CONSOLE	47
8.3. KDUMP USING RHEL SYSTEM ROLES	49

8.4. ADDITIONAL RESOURCES	49
CHAPTER 9. RECOVERING AND RESTORING A SYSTEM	50
9.1. SETTING UP REAR	50
CHAPTER 10. TROUBLESHOOTING PROBLEMS USING LOG FILES	52
10.1. SERVICES HANDLING SYSLOG MESSAGES	52
10.2. SUBDIRECTORIES STORING SYSLOG MESSAGES	52
10.3. INSPECTING LOG FILES USING THE WEB CONSOLE	52
10.4. VIEWING LOGS USING THE COMMAND LINE	53
10.5. ADDITIONAL RESOURCES	54
CHAPTER 11. ACCESSING THE RED HAT SUPPORT	55
11.1. OBTAINING RED HAT SUPPORT THROUGH RED HAT CUSTOMER PORTAL	55
11.2. TROUBLESHOOTING PROBLEMS USING SOSREPORT	55
CHAPTER 12. MANAGING SOFTWARE PACKAGES	57
12.1. SOFTWARE MANAGEMENT TOOLS IN RED HAT ENTERPRISE LINUX 8	57
12.2. APPLICATION STREAMS	57
12.3. SEARCHING FOR SOFTWARE PACKAGES	57
12.3.1. Searching packages with yum	58
12.3.2. Listing packages with yum	58
12.3.3. Listing repositories with yum	58
12.3.4. Displaying package information with yum	59
12.3.5. Listing package groups with yum	59
12.3.6. Specifying global expressions in yum input	59
12.4. INSTALLING SOFTWARE PACKAGES	60
12.4.1. Installing packages with yum	60
12.4.2. Installing a package group with yum	61
12.4.3. Specifying a package name in yum input	61
12.5. UPDATING SOFTWARE PACKAGES	62
12.5.1. Checking for updates with yum	62
12.5.2. Updating a single package with yum	62
12.5.3. Updating a package group with yum	62
12.5.4. Updating all packages and their dependencies with yum	62
12.5.5. Updating security-related packages with yum	63
12.5.6. Automating software updates	63
12.5.6.1. Installing DNF Automatic	63
12.5.6.2. DNF Automatic configuration file	63
12.5.6.3. Enabling DNF Automatic	64
12.5.6.4. Overview of the systemd timer units included in the dnf-automatic package	66
12.6. UNINSTALLING SOFTWARE PACKAGES	67
12.6.1. Removing packages with yum	67
12.6.2. Removing a package group with yum	68
12.6.3. Specifying a package name in yum input	68
12.7. MANAGING SOFTWARE PACKAGE GROUPS	68
12.7.1. Listing package groups with yum	69
12.7.2. Installing a package group with yum	69
12.7.3. Removing a package group with yum	69
12.7.4. Specifying global expressions in yum input	70
12.8. HANDLING PACKAGE MANAGEMENT HISTORY	70
12.8.1. Listing transactions with yum	70
12.8.2. Reverting transactions with yum	71
12.8.3. Repeating transactions with yum	71

12.8.4. Specifying global expressions in yum input	71
12.9. MANAGING SOFTWARE REPOSITORIES	72
12.9.1. Setting yum repository options	72
12.9.2. Adding a yum repository	72
12.9.3. Enabling a yum repository	73
12.9.4. Disabling a yum repository	73
12.10. CONFIGURING YUM	73
12.10.1. Viewing the current yum configurations	74
12.10.2. Setting yum main options	74
12.10.3. Using yum plug-ins	74
12.10.3.1. Managing yum plug-ins	74
12.10.3.2. Enabling yum plug-ins	74
12.10.3.3. Disabling yum plug-ins	74
CHAPTER 13. INTRODUCTION TO SYSTEMD	76
13.1. SYSTEMD UNIT TYPES	76
13.2. SYSTEMD MAIN FEATURES	77
13.3. COMPATIBILITY CHANGES	78
13.4. ADDITIONAL RESOURCES (OR NEXT STEPS)	79
CHAPTER 14. MANAGING SYSTEM SERVICES WITH SYSTEMCTL	80
14.1. SERVICE UNIT MANAGEMENT WITH SYSTEMCTL	80
14.2. COMPARISON OF A SERVICE UTILITY WITH SYSTEMCTL	80
14.3. LISTING SYSTEM SERVICES	81
14.4. DISPLAYING SYSTEM SERVICE STATUS	82
14.5. POSITIVE AND NEGATIVE SERVICE DEPENDENCIES	84
14.6. STARTING A SYSTEM SERVICE	85
14.7. STOPPING A SYSTEM SERVICE	85
14.8. RESTARTING A SYSTEM SERVICE	86
14.9. ENABLING A SYSTEM SERVICE	87
14.10. DISABLING A SYSTEM SERVICE	88
CHAPTER 15. WORKING WITH SYSTEMD TARGETS	90
15.1. DIFFERENCE BETWEEN SYSV RUNLEVELS AND SYSTEMD TARGETS	90
15.2. VIEWING THE DEFAULT TARGET	91
15.2.1. Changing the default target	92
15.2.2. Changing the default target using symbolic link	92
15.2.2.1. Booting to rescue mode	93
15.2.2.2. Booting to emergency mode	94
CHAPTER 16. SHUTTING DOWN, SUSPENDING, AND HIBERNATING THE SYSTEM	95
16.1. SYSTEM SHUTDOWN	95
16.2. SHUTTING DOWN THE SYSTEM USING THE SHUTDOWN COMMAND	95
16.3. SHUTTING DOWN THE SYSTEM USING THE SYSTEMCTL COMMAND	96
16.4. RESTARTING THE SYSTEM	96
16.5. SUSPENDING THE SYSTEM	97
16.6. HIBERNATING THE SYSTEM	97
16.7. OVERVIEW OF THE POWER MANAGEMENT COMMANDS WITH SYSTEMCTL	98
CHAPTER 17. WORKING WITH SYSTEMD UNIT FILES	99
17.1. INTRODUCTION TO UNIT FILES	99
17.2. UNIT FILE STRUCTURE	99
17.3. IMPORTANT [UNIT] SECTION OPTIONS	100
17.4. IMPORTANT SERVICE SECTION OPTIONS	101

17.5. IMPORTANT [INSTALL] SECTION OPTIONS	102
17.6. CREATING CUSTOM UNIT FILES	102
17.7. CREATING A CUSTOM UNIT FILE BY USING THE SECOND INSTANCE OF THE SSHD SERVICE	104
17.8. CONVERTING SYSV INIT SCRIPTS TO UNIT FILES	105
17.9. FINDING THE SYSTEMD SERVICE DESCRIPTION	106
17.10. FINDING THE SYSTEMD SERVICE DEPENDENCIES	106
17.11. FINDING DEFAULT TARGETS OF THE SERVICE	107
17.12. FINDING FILES USED BY THE SERVICE	107
17.13. MODIFYING EXISTING UNIT FILES	108
17.14. EXTENDING THE DEFAULT UNIT CONFIGURATION	109
17.15. OVERRIDING THE DEFAULT UNIT CONFIGURATION	111
17.16. CHANGING THE TIMEOUT LIMIT	111
17.17. MONITORING OVERRIDDEN UNITS	112
17.18. WORKING WITH INSTANTIATED UNITS	113
17.19. IMPORTANT UNIT SPECIFIERS	113
17.20. ADDITIONAL RESOURCES	114
CHAPTER 18. OPTIMIZING SYSTEMD TO SHORTEN THE BOOT TIME	115
18.1. EXAMINING SYSTEM BOOT PERFORMANCE	115
Analyzing overall boot time	115
Analyzing unit initialization time	115
Identifying critical units	115
18.2. A GUIDE TO SELECTING SERVICES THAT CAN BE SAFELY DISABLED	116
18.3. ADDITIONAL RESOURCES	120
CHAPTER 19. INTRODUCTION TO MANAGING USER AND GROUP ACCOUNTS	121
19.1. INTRODUCTION TO USERS AND GROUPS	121
19.2. CONFIGURING RESERVED USER AND GROUP IDS	121
19.3. USER PRIVATE GROUPS	122
CHAPTER 20. MANAGING USER ACCOUNTS IN THE WEB CONSOLE	123
20.1. SYSTEM USER ACCOUNTS MANAGED IN THE WEB CONSOLE	123
20.2. ADDING NEW ACCOUNTS USING THE WEB CONSOLE	123
20.3. ENFORCING PASSWORD EXPIRATION IN THE WEB CONSOLE	124
20.4. TERMINATING USER SESSIONS IN THE WEB CONSOLE	125
CHAPTER 21. MANAGING USERS FROM THE COMMAND LINE	126
21.1. ADDING A NEW USER FROM THE COMMAND LINE	126
21.2. ADDING A NEW GROUP FROM THE COMMAND LINE	126
21.3. ADDING A USER TO A SUPPLEMENTARY GROUP FROM THE COMMAND LINE	127
21.4. CREATING A GROUP DIRECTORY	128
CHAPTER 22. EDITING USER GROUPS USING THE COMMAND LINE	130
22.1. PRIMARY AND SUPPLEMENTARY USER GROUPS	130
22.2. LISTING THE PRIMARY AND SUPPLEMENTARY GROUPS OF A USER	130
22.3. CHANGING THE PRIMARY GROUP OF A USER	131
22.4. ADDING A USER TO A SUPPLEMENTARY GROUP FROM THE COMMAND LINE	132
22.5. REMOVING A USER FROM A SUPPLEMENTARY GROUP	132
22.6. CHANGING ALL OF THE SUPPLEMENTARY GROUPS OF A USER	133
CHAPTER 23. MANAGING SUDO ACCESS	135
23.1. USER AUTHORIZATIONS IN SUDOERS	135
23.2. GRANTING SUDO ACCESS TO A USER	136
23.3. ENABLING UNPRIVILEGED USERS TO RUN CERTAIN COMMANDS	137
23.4. ADDITIONAL RESOURCES	139

CHAPTER 24. CHANGING AND RESETTING THE ROOT PASSWORD	140
24.1. CHANGING THE ROOT PASSWORD AS THE ROOT USER	140
24.2. CHANGING OR RESETTING THE FORGOTTEN ROOT PASSWORD AS A NON-ROOT USER	140
24.3. RESETTING THE ROOT PASSWORD ON BOOT	140
CHAPTER 25. MANAGING FILE PERMISSIONS	143
25.1. BASE FILE PERMISSIONS	143
25.2. USER FILE-CREATION MODE MASK	145
25.3. DEFAULT FILE PERMISSIONS	146
25.4. CHANGING FILE PERMISSIONS USING SYMBOLIC VALUES	148
25.5. CHANGING FILE PERMISSIONS USING OCTAL VALUES	150
CHAPTER 26. MANAGING THE UMASK	151
26.1. DISPLAYING THE CURRENT VALUE OF THE UMASK	151
26.2. DISPLAYING THE DEFAULT BASH UMASK	151
26.3. SETTING THE UMASK USING SYMBOLIC VALUES	152
26.4. SETTING THE UMASK USING OCTAL VALUES	153
26.5. CHANGING THE DEFAULT UMASK FOR THE NON-LOGIN SHELL	153
26.6. CHANGING THE DEFAULT UMASK FOR THE LOGIN SHELL	154
26.7. CHANGING THE DEFAULT UMASK FOR A SPECIFIC USER	154
26.8. SETTING DEFAULT UMASK FOR NEWLY CREATED HOME DIRECTORIES	154
CHAPTER 27. USING DNSTAP IN RHEL	156
27.1. RECORDING DNS QUERIES USING DNSTAP IN RHEL	156
CHAPTER 28. MANAGING THE ACCESS CONTROL LIST	158
28.1. DISPLAYING THE CURRENT ACCESS CONTROL LIST	158
28.2. SETTING THE ACCESS CONTROL LIST	158
CHAPTER 29. USING THE CHRONY SUITE TO CONFIGURE NTP	160
29.1. INTRODUCTION TO CHRONY SUITE	160
29.2. USING CHRONYC TO CONTROL CHRONYD	160
29.3. MIGRATING TO CHRONY	161
29.3.1. Migration script	162
CHAPTER 30. USING CHRONY	163
30.1. MANAGING CHRONY	163
30.2. CHECKING IF CHRONY IS SYNCHRONIZED	163
30.3. MANUALLY ADJUSTING THE SYSTEM CLOCK	164
30.4. SETTING UP CHRONY FOR A SYSTEM IN AN ISOLATED NETWORK	165
30.5. ADDITIONAL RESOURCES	166
CHAPTER 31. CHRONY WITH HW TIMESTAMPING	167
31.1. VERIFYING SUPPORT FOR HARDWARE TIMESTAMPING	167
31.2. ENABLING HARDWARE TIMESTAMPING	168
31.3. CONFIGURING CLIENT POLLING INTERVAL	168
31.4. ENABLING INTERLEAVED MODE	168
31.5. CONFIGURING SERVER FOR LARGE NUMBER OF CLIENTS	169
31.6. VERIFYING HARDWARE TIMESTAMPING	169
31.7. CONFIGURING PTP-NTP BRIDGE	170
CHAPTER 32. ACHIEVING SOME SETTINGS PREVIOUSLY SUPPORTED BY NTP IN CHRONY	171
32.1. MONITORING BY NTPQ AND NTPDC	171
32.2. USING AUTHENTICATION MECHANISM BASED ON PUBLIC KEY CRYPTOGRAPHY	171
32.3. USING EPHEMERAL SYMMETRIC ASSOCIATIONS	172

32.4. MULTICAST/BROADCAST CLIENT	172
CHAPTER 33. USING SECURE COMMUNICATIONS BETWEEN TWO SYSTEMS WITH OPENSSH	173
33.1. SSH AND OPENSSH	173
33.2. CONFIGURING AND STARTING AN OPENSSH SERVER	174
33.3. SETTING AN OPENSSH SERVER FOR KEY-BASED AUTHENTICATION	175
33.4. GENERATING SSH KEY PAIRS	176
33.5. USING SSH KEYS STORED ON A SMART CARD	178
33.6. MAKING OPENSSH MORE SECURE	179
33.7. CONNECTING TO A REMOTE SERVER USING AN SSH JUMP HOST	181
33.8. CONNECTING TO REMOTE MACHINES WITH SSH KEYS USING SSH-AGENT	182
33.9. ADDITIONAL RESOURCES	183
CHAPTER 34. CONFIGURING A REMOTE LOGGING SOLUTION	185
34.1. THE RSYSLOG LOGGING SERVICE	185
34.2. INSTALLING RSYSLOG DOCUMENTATION	185
34.3. CONFIGURING A SERVER FOR REMOTE LOGGING OVER TCP	186
34.4. CONFIGURING REMOTE LOGGING TO A SERVER OVER TCP	188
34.5. CONFIGURING A SERVER FOR RECEIVING REMOTE LOGGING INFORMATION OVER UDP	189
34.6. CONFIGURING REMOTE LOGGING TO A SERVER OVER UDP	191
34.7. CONFIGURING RELIABLE REMOTE LOGGING	192
34.8. SUPPORTED RSYSLOG MODULES	194
34.9. ADDITIONAL RESOURCES	194
CHAPTER 35. USING THE LOGGING SYSTEM ROLE	195
35.1. THE LOGGING SYSTEM ROLE	195
35.2. LOGGING SYSTEM ROLE PARAMETERS	195
35.3. APPLYING A LOCAL LOGGING SYSTEM ROLE	196
35.4. FILTERING LOGS IN A LOCAL LOGGING SYSTEM ROLE	198
35.5. APPLYING A REMOTE LOGGING SOLUTION USING THE LOGGING SYSTEM ROLE	200
35.6. ADDITIONAL RESOURCES	203
CHAPTER 36. USING THE LOGGING SYSTEM ROLES WITH RELP	204
36.1. CONFIGURING CLIENT LOGGING WITH RELP	204
36.2. CONFIGURING SERVER LOGGING WITH RELP	206
CHAPTER 37. USING THE LOGGING SYSTEM ROLE WITH TLS	209
37.1. CONFIGURING CLIENT LOGGING WITH TLS	209
37.2. CONFIGURING SERVER LOGGING WITH TLS	210
CHAPTER 38. INTRODUCTION TO PYTHON	213
38.1. PYTHON VERSIONS	213
CHAPTER 39. INSTALLING AND USING PYTHON	215
39.1. INSTALLING PYTHON 3	215
39.2. INSTALLING ADDITIONAL PYTHON 3 PACKAGES	216
39.3. INSTALLING ADDITIONAL PYTHON 3 TOOLS FOR DEVELOPERS	216
39.4. INSTALLING PYTHON 2	217
39.5. MIGRATING FROM PYTHON 2 TO PYTHON 3	218
39.6. USING PYTHON	218
CHAPTER 40. CONFIGURING THE UNVERSIONED PYTHON	220
40.1. CONFIGURING THE UNVERSIONED PYTHON COMMAND DIRECTLY	220
40.2. CONFIGURING THE UNVERSIONED PYTHON COMMAND TO THE REQUIRED PYTHON VERSION INTERACTIVELY	220

40.3. ADDITIONAL RESOURCES	221
CHAPTER 41. PACKAGING PYTHON 3 RPMS	222
41.1. SPEC FILE DESCRIPTION FOR A PYTHON PACKAGE	222
41.2. COMMON MACROS FOR PYTHON 3 RPMS	224
41.3. AUTOMATIC PROVIDES FOR PYTHON RPMS	224
CHAPTER 42. HANDLING INTERPRETER DIRECTIVES IN PYTHON SCRIPTS	225
42.1. MODIFYING INTERPRETER DIRECTIVES IN PYTHON SCRIPTS	225
42.2. CHANGING /USR/BIN/PYTHON3 INTERPRETER DIRECTIVES IN YOUR CUSTOM PACKAGES	226
CHAPTER 43. USING THE PHP SCRIPTING LANGUAGE	227
43.1. INSTALLING THE PHP SCRIPTING LANGUAGE	227
43.2. USING THE PHP SCRIPTING LANGUAGE WITH A WEB SERVER	228
43.2.1. Using PHP with the Apache HTTP Server	228
43.2.2. Using PHP with the nginx web server	229
43.3. RUNNING A PHP SCRIPT USING THE COMMAND-LINE INTERFACE	231
43.4. ADDITIONAL RESOURCES	232
CHAPTER 44. USING LANGPACKS	233
44.1. CHECKING LANGUAGES THAT PROVIDE LANGPACKS	233
44.2. WORKING WITH RPM WEAK DEPENDENCY-BASED LANGPACKS	233
44.2.1. Listing already installed language support	233
44.2.2. Checking the availability of language support	234
44.2.3. Listing packages installed for a language	234
44.2.4. Installing language support	234
44.2.5. Removing language support	234
44.3. SAVING DISK SPACE BY USING GLIBC-LANGPACK-<LOCALE_CODE>	234
CHAPTER 45. GETTING STARTED WITH TCL/TK	236
45.1. INTRODUCTION TO TCL/TK	236
45.2. NOTABLE CHANGES IN TCL/TK 8.6	236
45.3. MIGRATING TO TCL/TK 8.6	237
45.3.1. Migration path for developers of Tcl extensions	237
45.3.2. Migration path for users scripting their tasks with Tcl/Tk	237

MAKING OPEN SOURCE MORE INCLUSIVE

Red Hat is committed to replacing problematic language in our code, documentation, and web properties. We are beginning with these four terms: master, slave, blacklist, and whitelist. Because of the enormity of this endeavor, these changes will be implemented gradually over several upcoming releases. For more details, see [our CTO Chris Wright's message](#).

PROVIDING FEEDBACK ON RED HAT DOCUMENTATION

We appreciate your input on our documentation. Please let us know how we could make it better. To do so:

- For simple comments on specific passages:
 1. Make sure you are viewing the documentation in the *Multi-page HTML* format. In addition, ensure you see the **Feedback** button in the upper right corner of the document.
 2. Use your mouse cursor to highlight the part of text that you want to comment on.
 3. Click the **Add Feedback** pop-up that appears below the highlighted text.
 4. Follow the displayed instructions.
- For submitting more complex feedback, create a Bugzilla ticket:
 1. Go to the [Bugzilla](#) website.
 2. As the Component, use **Documentation**.
 3. Fill in the **Description** field with your suggestion for improvement. Include a link to the relevant part(s) of documentation.
 4. Click **Submit Bug**.

CHAPTER 1. GETTING STARTED WITH RHEL SYSTEM ROLES

This section explains what RHEL System Roles are. Additionally, it describes how to apply a particular role through an Ansible playbook to perform various system administration tasks.

1.1. INTRODUCTION TO RHEL SYSTEM ROLES

RHEL System Roles is a collection of Ansible roles and modules. RHEL System Roles 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.

On Red Hat Enterprise Linux 8, the interface currently consists of the following roles:

- `kdump`
- `network`
- `selinux`
- `storage`
- `certificate`
- `kernel_settings`
- `logging`
- `metrics`
- `nbde_client` and `nbde_server`
- `timesync`
- `tlog`

All these roles are provided by the **rhel-system-roles** package available in the **AppStream** repository.

Additional resources

- [Red Hat Enterprise Linux \(RHEL\) System Roles](#)
- `/usr/share/doc/rhel-system-roles` documentation ^[1]
- [Introduction to the SELinux system role](#)
- [Introduction to the storage role](#)
- [Administration and configuration tasks using System Roles in RHEL](#)

1.2. RHEL SYSTEM ROLES TERMINOLOGY

You can find the following terms across this documentation:

System Roles terminology

Ansible playbook

Playbooks are Ansible's configuration, deployment, and orchestration language. They can describe a policy you want your remote systems to enforce, or a set of steps in a general IT process.

Control node

Any machine with Ansible installed. You can run commands and playbooks, invoking `/usr/bin/ansible` or `/usr/bin/ansible-playbook`, from any control node. You can use any computer that has Python installed on it as a control node – laptops, shared desktops, and servers can all run Ansible. However, you cannot use a Windows machine as a control node. You can have multiple control nodes.

Inventory

A list of managed nodes. An inventory file is also sometimes called a "hostfile". Your inventory can specify information like IP address for each managed node. An inventory can also organize managed nodes, creating and nesting groups for easier scaling. To learn more about inventory, see the [Working with Inventory](#) section.

Managed nodes

The network devices, servers, or both that you manage with Ansible. Managed nodes are also sometimes called "hosts". Ansible is not installed on managed nodes.

1.3. APPLYING A ROLE

The following procedure describes how to apply a particular role.

Prerequisites

- Ensure that the **rhel-system-roles** package is installed on the system that you want to use as a control node:

```
# yum install rhel-system-roles
```
- You need the **ansible** package to run playbooks that use RHEL System Roles. Ensure that the Ansible Engine repository is enabled, and the **ansible** package is installed on the system that you want to use as a control node.
 - 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?](#).
- Ensure that you are able to create an Ansible inventory. Inventories represent the hosts, host groups, and some of the configuration parameters used by the Ansible playbooks.

Playbooks are typically human-readable, and are defined in **ini**, **yaml**, **json**, and other file formats.

- Ensure that 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 are defined in the **yaml** format.

Procedure

1. Create the required Ansible inventory containing the hosts and groups that you want to manage. Here is an example using a file called **inventory.ini** of a group of hosts called **webservers**:

```
[webservers]
host1
host2
host3
```

2. Create an Ansible playbook including the required role. The following example shows how to use roles through the **roles:** option for a playbook:

The following example shows how to use roles through the **roles:** option for a given **play**:

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

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/SUBSYSTEM/
```

Replace *SUBSYSTEM* with the name of the required role, such as **selinux**, **kdump**, **network**, **timesync**, or **storage**.

3. To execute the playbook on specific hosts, you must perform one of the following:
 - Edit the playbook to use **hosts: host1[,host2,...]**, or **hosts: all**, and execute the command:

```
# ansible-playbook name.of.the.playbook
```

- Edit the inventory to ensure that the hosts you want to use are defined in a group, and execute the command:

```
# ansible-playbook -i name.of.the.inventory name.of.the.playbook
```

- Specify all hosts when executing the **ansible-playbook** command:

```
# ansible-playbook -i host1,host2,... name.of.the.playbook
```



IMPORTANT

Be aware that the **-i** flag specifies the inventory of all hosts that are available. If you have multiple targeted hosts, but want to select a host against which you want to run the playbook, you can add a variable in the playbook to be able to select a host. For example:

Ansible Playbook | `example-playbook.yml`:

```
- hosts: "{{ target_host }}"
  roles:
    - rhel-system-roles.network
    - rhel-system-roles.timesync
```

Playbook execution command:

```
# ansible-playbook -i host1,...hostn -e target_host=host5 example-playbook.yml
```

Additional resources

- [Ansible playbooks](#)
- [Using roles in Ansible playbook](#)
- [Examples of Ansible playbooks](#)
- [How to create and work with inventory?](#)
- [ansible-playbook](#)

1.4. ADDITIONAL RESOURCES

- [Red Hat Enterprise Linux \(RHEL\) System Roles Red Hat Knowledgebase article](#)
- [Managing local storage using RHEL System Roles](#)
- [Deploying the same SELinux configuration on multiple systems using RHEL System Roles](#)

[1] This documentation is installed automatically with the **rhel-system-roles** package.

CHAPTER 2. CHANGING BASIC ENVIRONMENT SETTINGS

Configuration of basic environment settings is a part of the installation process. The following sections guide you when you change them later. The basic configuration of the environment includes:

- Date and time
- System locales
- Keyboard layout
- Language

2.1. 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, see [Using the chrony suite to configure NTP](#).

2.1.1. Displaying the current date and time

To display the current date and time, use either of these steps.

Procedure

1. Enter the **date** command:

```
$ date
Mon Mar 30 16:02:59 CEST 2020
```

2. To see more details, use the **timedatectl** command:

```
$ timedatectl
Local time: Mon 2020-03-30 16:04:42 CEST
Universal time: Mon 2020-03-30 14:04:42 UTC
RTC time: Mon 2020-03-30 14:04:41
Time zone: Europe/Prague (CEST, +0200)
System clock synchronized: yes
NTP service: active
RTC in local TZ: no
```

Additional resources

- [Configuring time settings using the web console](#)
- **man date(1)** and **man timedatectl(1)**

2.2. 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. Every service or user inherits the locale settings configured in **/etc/locale.conf**, unless individual programs or individual users override them.

This section describes how to manage system locale.

Procedure

- To list available system locale settings:

```
$ localectl list-locales
C.utf8
aa_DJ
aa_DJ.iso88591
aa_DJ.utf8
...
```

- To display the current status of the system locales settings:

```
$ localectl status
```

- To set or change the default system locale settings, use a **localectl set-locale** sub-command as the **root** user. For example:

```
# localectl set-locale LANG=en-US
```

Additional resources

- **man localectl(1)**, **man locale(7)**, and **man locale.conf(5)**

2.3. CONFIGURING THE KEYBOARD LAYOUT

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

Procedure

- To list available keymaps:

```
$ localectl list-keymaps
ANSI-dvorak
al
al-plisi
amiga-de
amiga-us
...
```

- To display the current status of keymaps settings:

```
$ localectl status
...
VC Keymap: us
...
```

- To set or change the default system keymap. For example:

```
# localectl set-keymap us
```

Additional resources

- **man localectl(1)**, **man locale(7)**, and **man locale.conf(5)**

2.4. CHANGING THE LANGUAGE USING DESKTOP GUI

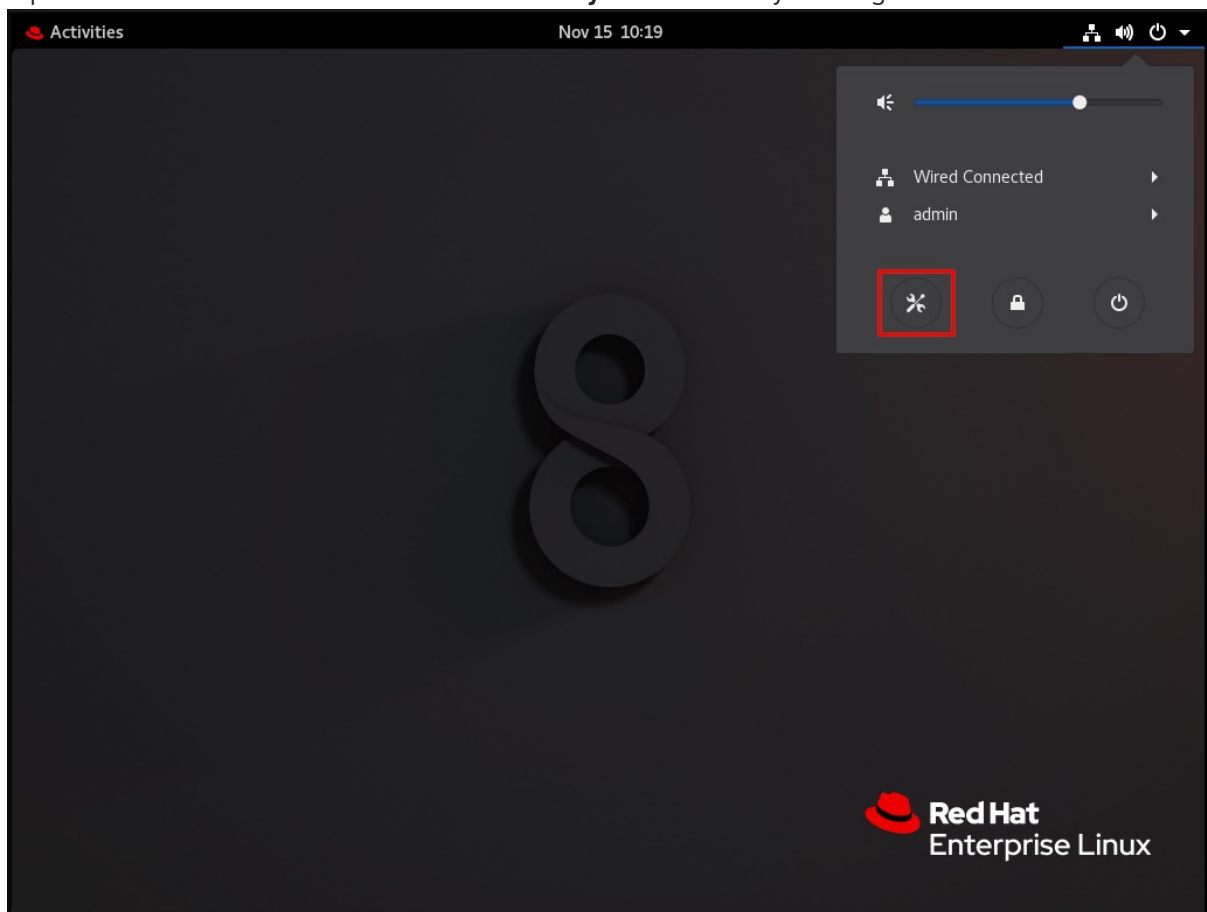
This section describes how to change the system language using the desktop GUI.

Prerequisites

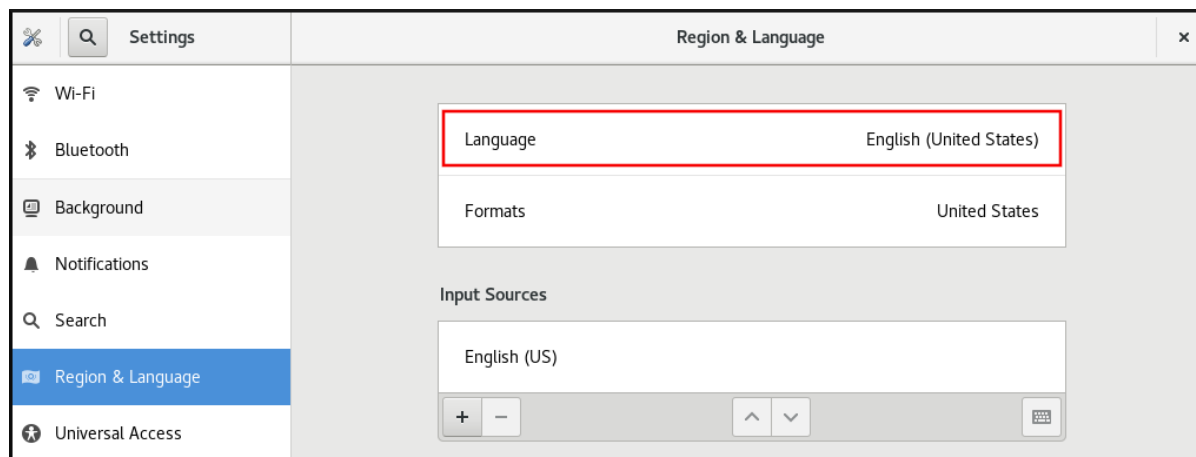
- Required language packages are installed on your system

Procedure

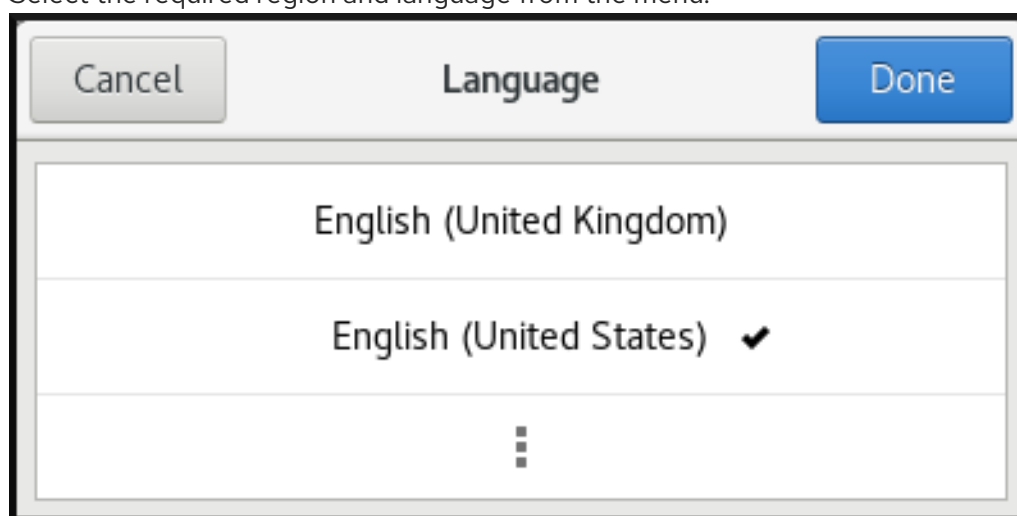
1. Open the **GNOME Control Center** from the **System menu** by clicking on its icon.



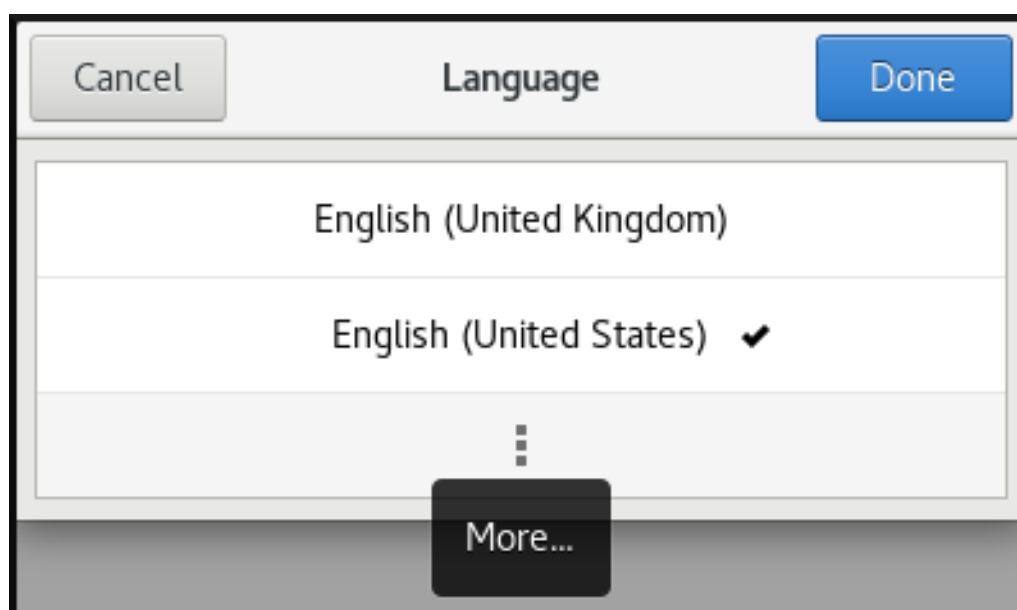
2. In the **GNOME Control Center**, choose **Region & Language** from the left vertical bar.
3. Click the **Language** menu.



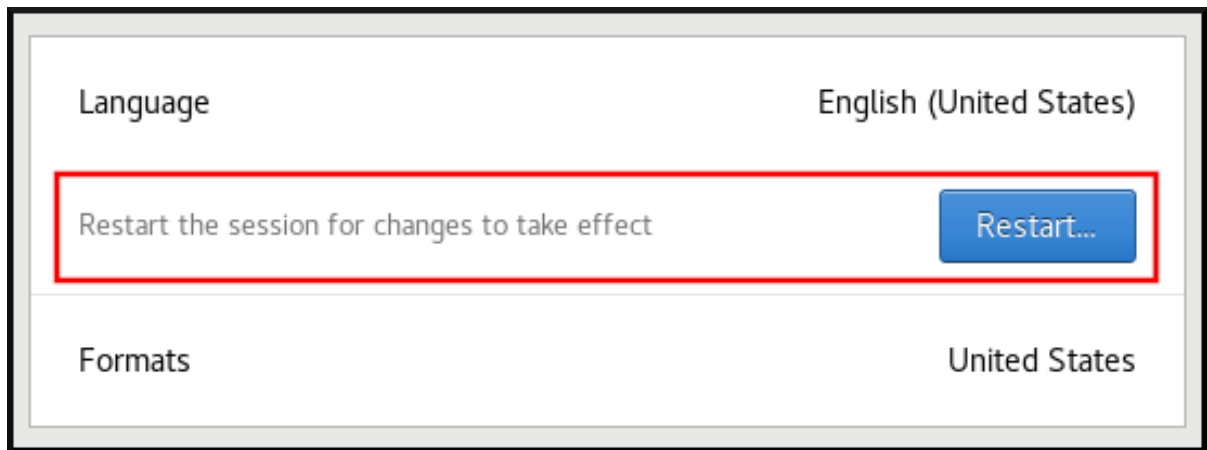
4. Select the required region and language from the menu.



If your region and language are not listed, scroll down, and click **More** to select from available regions and languages.



5. Click **Done**.
6. Click **Restart** for changes to take effect.

**NOTE**

Some applications do not support certain languages. The text of an application that cannot be translated into the selected language remains in US English.

Additional resources

- [Launching applications in GNOME](#)

2.5. ADDITIONAL RESOURCES

- [Performing a standard RHEL installation](#)

CHAPTER 3. CONFIGURING AND MANAGING NETWORK ACCESS

This section describes different options on how to add Ethernet connections in Red Hat Enterprise Linux.

3.1. CONFIGURING THE NETWORK AND HOST NAME IN THE GRAPHICAL INSTALLATION MODE

Follow the steps in this procedure to configure your network and host name.

Procedure

1. From the **Installation Summary** window, click **Network and Host Name**.
2. From the list in the left-hand pane, select an interface. The details are displayed in the right-hand pane.



NOTE

- There are several types of network device naming standards used to identify network devices with persistent names, for example, **em1** and **wl3sp0**. For information about these standards, see the [Configuring and managing networking](#) document.

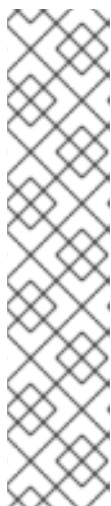
3. Toggle the **ON/OFF** switch to enable or disable the selected interface.



NOTE

The installation program automatically detects locally accessible interfaces, and you cannot add or remove them manually.

4. Click **+** to add a virtual network interface, which can be either: Team, Bond, Bridge, or VLAN.
5. Click **-** to remove a virtual interface.
6. Click **Configure** to change settings such as IP addresses, DNS servers, or routing configuration for an existing interface (both virtual and physical).
7. Type a host name for your system in the **Host Name** field.



NOTE

- The host name can be either a fully-qualified domain name (FQDN) in the format *hostname.domainname*, or a short host name with no domain name. Many networks have a Dynamic Host Configuration Protocol (DHCP) service that automatically supplies connected systems with a domain name. To allow the DHCP service to assign the domain name to this machine, specify only the short host name. The value **localhost** means that no specific static host name for the target system is configured, and the actual host name of the installed system is configured during the processing of the network configuration, for example, by **NetworkManager** using DHCP or DNS.
- Host names can only contain alpha-numeric characters and - or .. Host names cannot start or end with - and ..

8. Click **Apply** to apply the host name to the installer environment.
9. Alternatively, in the **Network and Hostname** window, you can choose the Wireless option. Click **Select network** in the right-hand pane to select your wifi connection, enter the password if required, and click **Done**.

Additional resources

- [Performing an advanced RHEL installation](#) .

3.2. CONFIGURING A STATIC ETHERNET CONNECTION USING NMCLI

This procedure describes adding an Ethernet connection with the following settings using the **nmcli** utility:

- A static IPv4 address - **192.0.2.1** with a **/24** subnet mask
- A static IPv6 address - **2001:db8:1::1** with a **/64** subnet mask
- An IPv4 default gateway - **192.0.2.254**
- An IPv6 default gateway - **2001:db8:1::fffe**
- An IPv4 DNS server - **192.0.2.200**
- An IPv6 DNS server - **2001:db8:1::ffbb**
- A DNS search domain - **example.com**

Procedure

1. Add a new NetworkManager connection profile for the Ethernet connection:

```
# nmcli connection add con-name Example-Connection ifname enp7s0 type ethernet
```

The further steps modify the **Example-Connection** connection profile you created.

2. Set the IPv4 address:

```
# nmcli connection modify Example-Connection ipv4.addresses 192.0.2.1/24
```

3. Set the IPv6 address:

```
# nmcli connection modify Example-Connection ipv6.addresses 2001:db8:1::1/64
```

4. Set the IPv4 and IPv6 connection method to **manual**:

```
# nmcli connection modify Example-Connection ipv4.method manual
# nmcli connection modify Example-Connection ipv6.method manual
```

5. Set the IPv4 and IPv6 default gateways:

```
# nmcli connection modify Example-Connection ipv4.gateway 192.0.2.254
# nmcli connection modify Example-Connection ipv6.gateway 2001:db8:1::fffe
```

6. Set the IPv4 and IPv6 DNS server addresses:

```
# nmcli connection modify Example-Connection ipv4.dns "192.0.2.200"
# nmcli connection modify Example-Connection ipv6.dns "2001:db8:1::ffbb"
```

To set multiple DNS servers, specify them space-separated and enclosed in quotes.

7. Set the DNS search domain for the IPv4 and IPv6 connection:

```
# nmcli connection modify Example-Connection ipv4.dns-search example.com
# nmcli connection modify Example-Connection ipv6.dns-search example.com
```

8. Activate the connection profile:

```
# nmcli connection up Example-Connection
Connection successfully activated (D-Bus active path:
/org/freedesktop/NetworkManager/ActiveConnection/13)
```

Verification steps

1. Display the status of the devices and connections:

```
# nmcli device status
DEVICE    TYPE    STATE    CONNECTION
enp7s0    ethernet connected Example-Connection
```

2. To display all settings of the connection profile:

```
# nmcli connection show Example-Connection
connection.id:      Example-Connection
connection.uuid:    b6cdfa1c-e4ad-46e5-af8b-a75f06b79f76
connection.stable-id: --
connection.type:    802-3-ethernet
connection.interface-name: enp7s0
...
```

3. Use the **ping** utility to verify that this host can send packets to other hosts.

- Ping an IP address in the same subnet.

For IPv4:

```
# ping 192.0.2.3
```

For IPv6:

```
# ping 2001:db8:2::1
```

If the command fails, verify the IP and subnet settings.

- Ping an IP address in a remote subnet.

For IPv4:

```
# ping 198.162.3.1
```

For IPv6:

```
# ping 2001:db8:2::1
```

- If the command fails, ping the default gateway to verify settings.

For IPv4:

```
# ping 192.0.2.254
```

For IPv6:

```
# ping 2001:db8:1::fffe
```

4. Use the **host** utility to verify that name resolution works. For example:

```
# host client.example.com
```

If the command returns any error, such as **connection timed out** or **no servers could be reached**, verify your DNS settings.

Troubleshooting steps

1. If the connection fails or if the network interface switches between an up and down status:
 - Make sure that the network cable is plugged-in to the host and a switch.
 - Check whether the link failure exists only on this host or also on other hosts connected to the same switch the server is connected to.
 - Verify that the network cable and the network interface are working as expected. Perform hardware diagnosis steps and replace defect cables and network interface cards.
 - If the configuration on the disk does not match the configuration on the device, starting or restarting NetworkManager creates an in-memory connection that reflects the configuration of the device. For further details and how to avoid this problem, see [NetworkManager duplicates a connection after restart of NetworkManager service](#)

Additional resources

- **nm-settings(5)**, **nmcli** and **nmcli(1)** man pages
- [Configuring NetworkManager to avoid using a specific profile to provide a default gateway](#)

3.3. ADDING A CONNECTION PROFILE USING NMTUI

The **nmtui** application provides a text user interface to NetworkManager. This procedure describes how to add a new connection profile.

Prerequisites

- The **NetworkManager-tui** package is installed.

Procedure

1. Start the NetworkManager text user interface utility:

```
# nmtui
```

2. Select the **Edit a connection** menu entry, and press **Enter**.
3. Select the **Add** button, and press **Enter**.
4. Select **Ethernet**, and press **Enter**.
5. Fill the fields with the connection details.

Edit Connection

Profile name enpls0
 Device enpls0 (52:54:00:DF:55:D1)

= ETHERNET <Show>

= IPv4 CONFIGURATION <Manual> <Hide>

Addresses 192.0.2.1/24
<Add...>
 Gateway 192.0.2.254
 DNS servers 192.0.2.254
<Add...>
 Search domains <Add...>

 Routing (No custom routes) <Edit...>
☐ Never use this network for default route
☐ Ignore automatically obtained routes
☐ Ignore automatically obtained DNS parameters

☐ Require IPv4 addressing for this connection

<Remove>
<Remove>

= IPv6 CONFIGURATION <Manual> <Hide>

Addresses 2001:db8:1::1/64
<Add...>
 Gateway 2001:db8:1::fffe
 DNS servers 2001:db8:1::fffe
<Add...>
 Search domains <Add...>

 Routing (No custom routes) <Edit...>
☐ Never use this network for default route
☐ Ignore automatically obtained routes
☐ Ignore automatically obtained DNS parameters

☐ Require IPv6 addressing for this connection

<Remove>
<Remove>

☒ Automatically connect
☒ Available to all users

<Cancel> <OK>

6. Select **OK** to save the changes.
7. Select **Back** to return to the main menu.
8. Select **Activate a connection**, and press **Enter**.
9. Select the new connection entry, and press **Enter** to activate the connection.
10. Select **Back** to return to the main menu.
11. Select **Quit**.

Verification steps

1. Display the status of the devices and connections:

nmcli device status

DEVICE	TYPE	STATE	CONNECTION
<i>enp1s0</i>	ethernet	connected	<i>Example-Connection</i>

2. To display all settings of the connection profile:

nmcli connection show Example-Connection

```
connection.id:      Example-Connection
connection.uuid:    b6cdfa1c-e4ad-46e5-af8b-a75f06b79f76
connection.stable-id:  --
connection.type:    802-3-ethernet
connection.interface-name: enp1s0
...
```

If the configuration on the disk does not match the configuration on the device, starting or restarting NetworkManager creates an in-memory connection that reflects the configuration of the device. For further details and how to avoid this problem, see [NetworkManager duplicates a connection after restart of NetworkManager service](#).

Additional resources

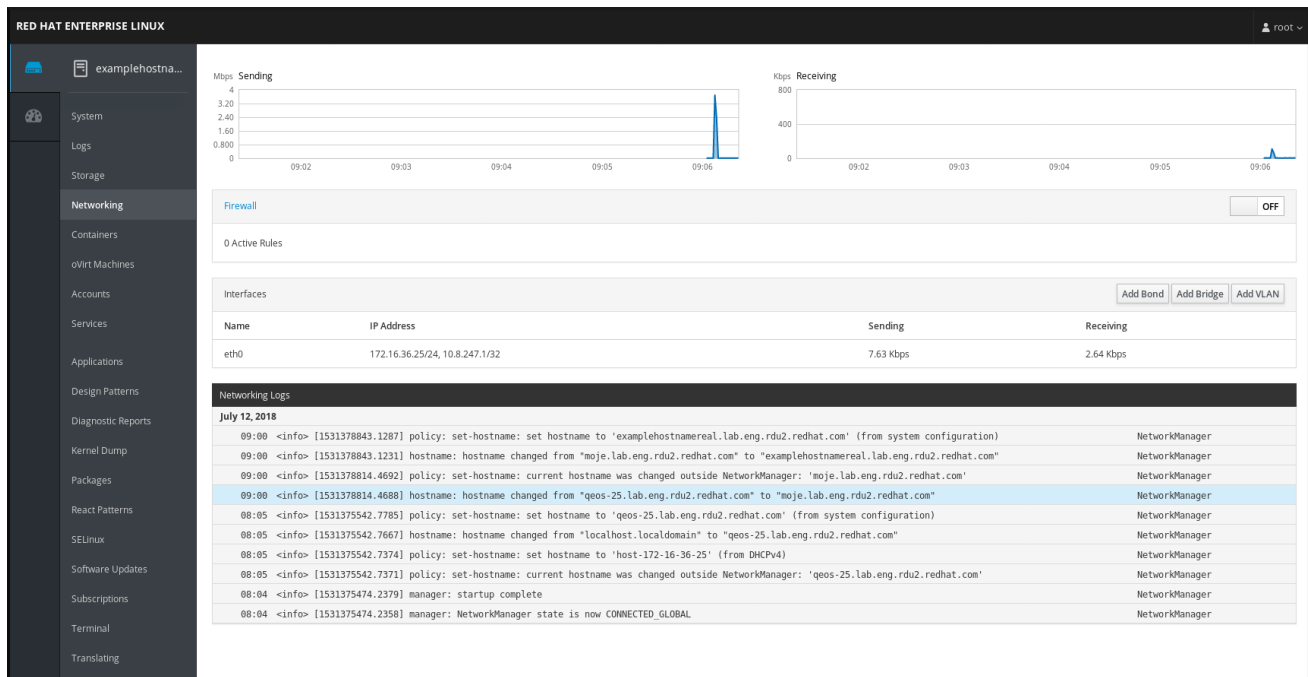
- [Testing basic network settings](#)
- **nmtui(1)** man page

3.4. MANAGING NETWORKING IN THE RHEL WEB CONSOLE

In the web console, the **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 3.1. Managing Networking in the RHEL web console



3.5. MANAGING NETWORKING USING RHEL SYSTEM ROLES

You can configure the networking connections on multiple target machines using the **network** role.

The **network** role allows to configure the following types of interfaces:

- Ethernet
- Bridge
- Bonded
- VLAN
- MacVLAN
- Infiniband

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:

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
```

Additional resources

- [Introduction to RHEL System Roles](#)

3.6. ADDITIONAL RESOURCES

- [Configuring and managing networking](#)

CHAPTER 4. REGISTERING THE SYSTEM AND MANAGING SUBSCRIPTIONS

Subscriptions cover products installed on Red Hat Enterprise Linux, including the operating system itself.

You can use a subscription to Red Hat Content Delivery Network to track:

- Registered systems
- Products installed on your systems
- Subscriptions attached to the installed products

4.1. REGISTERING THE SYSTEM AFTER THE INSTALLATION

Use the following procedure to register your system if you have not registered it during the installation process already.

Prerequisites

- A valid user account in the Red Hat Customer Portal.
- See the [Create a Red Hat Login](#) page.
- An active subscription for the RHEL system.
- For more information about the installation process, see [Performing a standard RHEL installation](#).

Procedure

1. Register and automatically subscribe your system in one step:

```
# subscription-manager register --username <username> --password <password> --auto-attach
Registering to: subscription.rhsm.redhat.com:443/subscription
The system has been registered with ID: 37to907c-ece6-49ea-9174-20b87ajk9ee7
The registered system name is: client1.idm.example.com
Installed Product Current Status:
Product Name: Red Hat Enterprise Linux for x86_64
Status:      Subscribed
```

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

If the registration process fails, you can register your system with a specific pool. For guidance on how to do it, proceed with the following steps:

- a. 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.

- b. 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
```

Additional resources

- [Understanding autoattaching subscriptions on the Customer Portal](#)
- [Understanding the manual registration and subscription on the Customer Portal](#)

4.2. REGISTERING SUBSCRIPTIONS WITH CREDENTIALS IN THE WEB CONSOLE

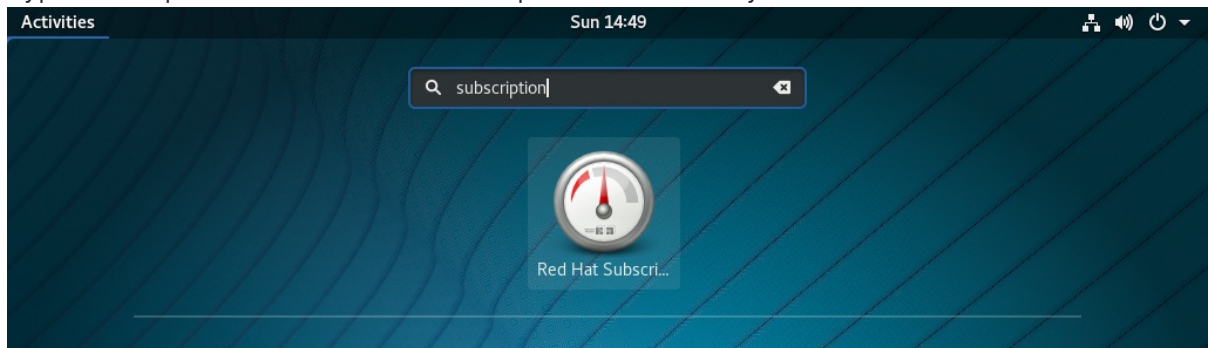
Use the following steps to register a newly installed Red Hat Enterprise Linux using the RHEL web console.

Prerequisites

- A valid user account on the Red Hat Customer Portal.
See the [Create a Red Hat Login](#) page.
- Active subscription for your RHEL system.

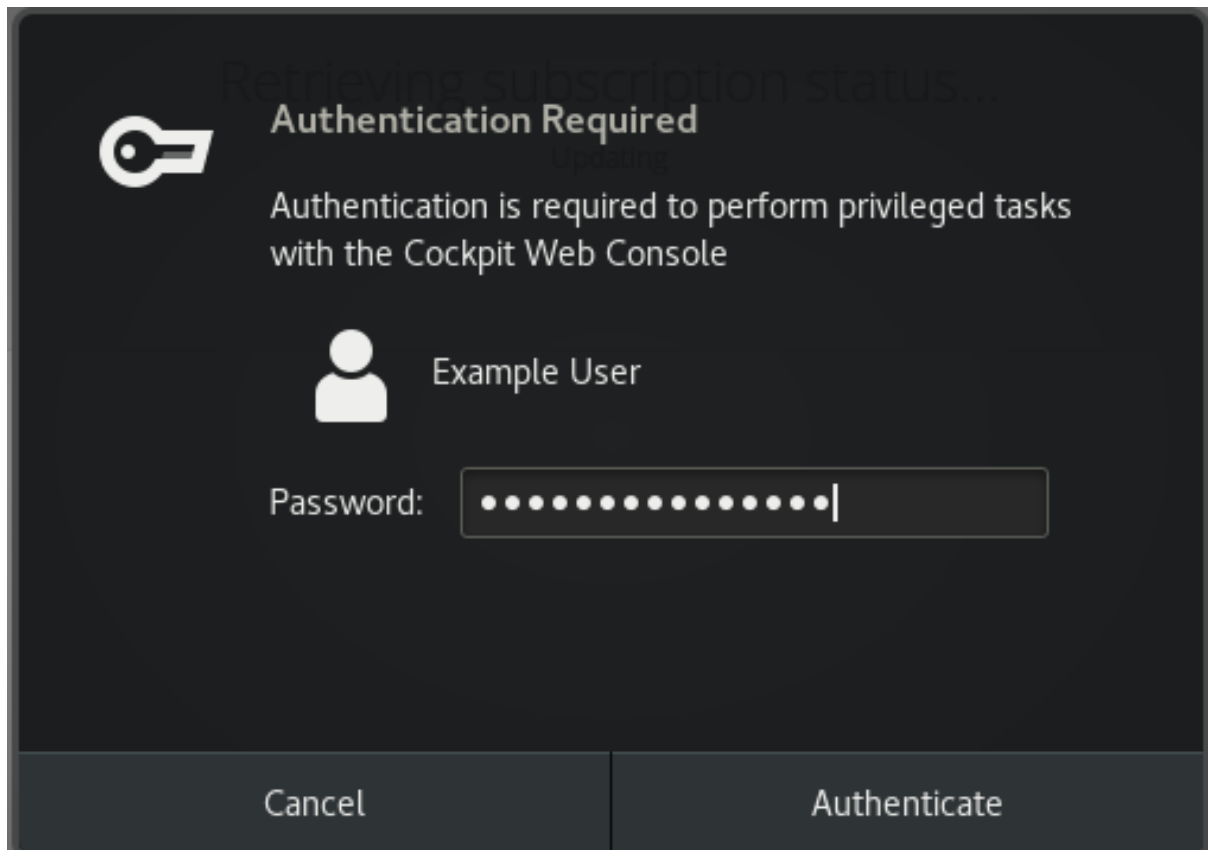
Procedure

1. Type **subscription** in the search field and press the **Enter** key.

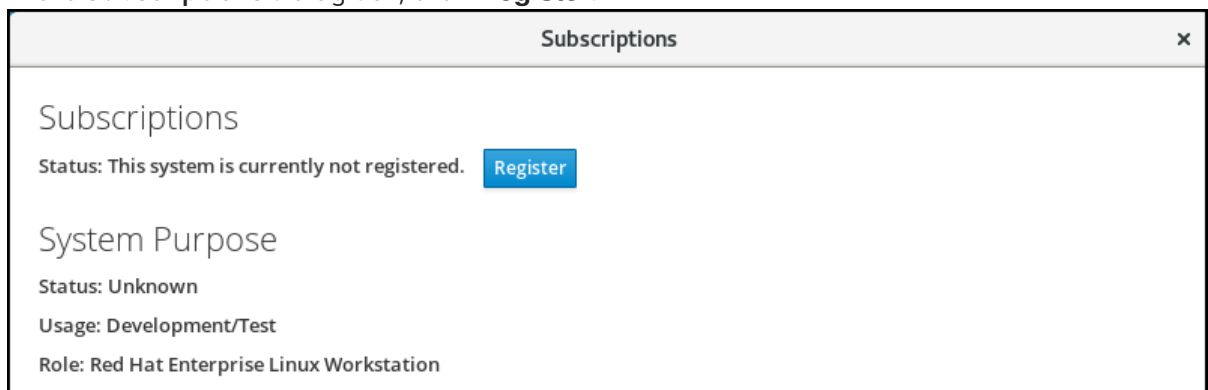


Alternatively, you can log in to the RHEL web console. For details, see [Logging in to the web console](#).

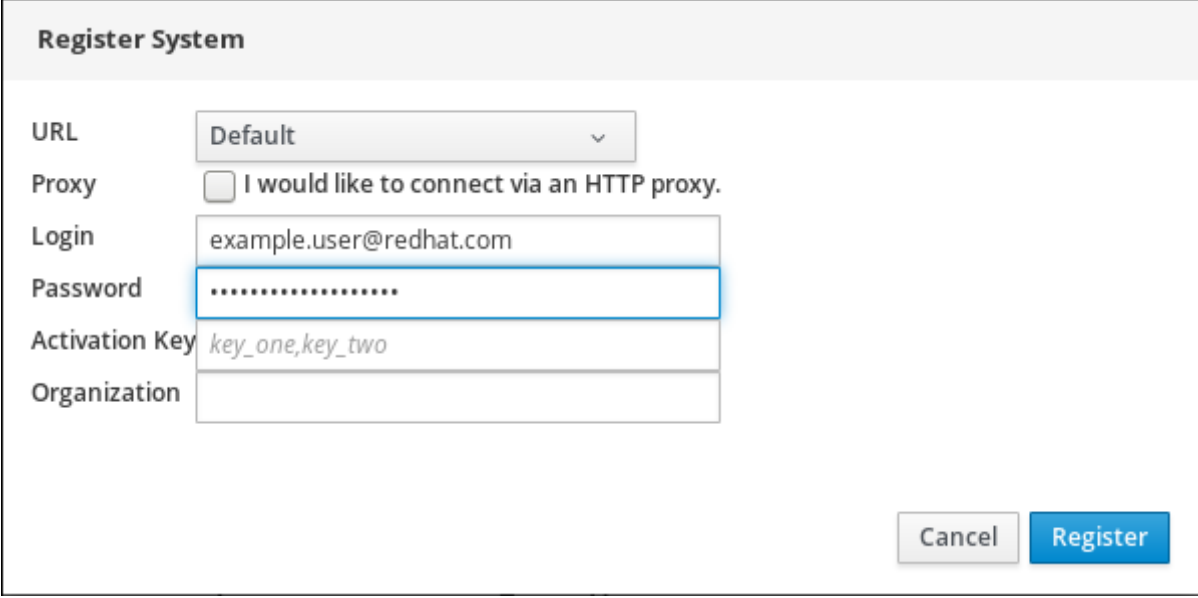
2. In the **polkit** authentication dialog for privileged tasks, add the password belonging to the user name displayed in the dialog.



3. Click **Authenticate**.
4. In the **Subscriptions** dialog box, click **Register**.



5. Enter your Customer Portal credentials.

A dialog box titled "Register System" with a light gray header. It contains several input fields: "URL" with a dropdown menu showing "Default"; "Proxy" with an unchecked checkbox and the text "I would like to connect via an HTTP proxy."; "Login" with a text field containing "example.user@redhat.com"; "Password" with a text field containing ten dots, which is highlighted with a blue border; "Activation Key" with a text field containing "key_one,key_two"; and "Organization" with an empty text field. At the bottom right, there are two buttons: "Cancel" and "Register".

URL	Default
Proxy	<input type="checkbox"/> I would like to connect via an HTTP proxy.
Login	example.user@redhat.com
Password
Activation Key	key_one,key_two
Organization	

Cancel Register

6. Enter the name of your organization.

If you have more than one account on the Red Hat Customer Portal, you have to add the organization name or organization ID. To get the org ID, go to your Red Hat contact point.

7. Click the **Register** button.

At this point, your Red Hat Enterprise Linux system has been successfully registered.

Subscriptions

Status: Current Unregister



System Purpose

Status: Unknown

Usage: Development/Test

Role: Red Hat Enterprise Linux Workstation

Installed products

Red Hat Enterprise Linux for x86_64 High Touch Beta

Product Name	Red Hat Enterprise Linux for x86_64 High Touch Beta
Product ID	230
Version	8.0 HTB
Arch	x86_64
Status	Subscribed
Starts	10/07/2018
Ends	10/06/2019

4.3. REGISTERING A SYSTEM USING RED HAT ACCOUNT ON GNOME

Follow the steps in this procedure to enroll your system with your Red Hat account.

Prerequisites

- A valid account on Red Hat customer portal.
See the [Create a Red Hat Login](#) page for new user registration.

Procedure

1. Go to the **system menu**, which is accessible from the top-right screen corner and click the **Settings** icon.
2. In the **Details** → **About** section, click **Register**.
3. Select **Registration Server**.
4. If you are not using the Red Hat server, enter the server address in the **URL** field.
5. In the **Registration Type** menu, select **Red Hat Account**

6. Under **Registration Details**:

- Enter your Red hat account user name in the **Login** field,
- Enter your Red hat account password in the **Password** field.
- Enter the name of your organization in the **Organization** field.

7. Click **Register**.

4.4. REGISTERING A SYSTEM USING AN ACTIVATION KEY ON GNOME

Follow the steps in this procedure to register your system with an activation key. You can get the activation key from your organization administrator.

Prerequisites

- Activation key or keys.
See the [Activation Keys](#) page for creating new activation keys.

Procedure

1. Go to the **system menu**, which is accessible from the top-right screen corner and click the **Settings** icon.
2. In the **Details** → **About** section, click **Register**.
3. Select **Registration Server**.
4. Enter **URL** to the customized server, if you are not using the Red Hat server.
5. In the **Registration Type** menu, select **Activation Keys**.
6. Under **Registration Details**:
 - Enter **Activation Keys**.
Separate multiple keys by a comma (,).
 - Enter the name or ID of your organization in the **Organization** field.
7. Click **Register**

4.5. REGISTERING RHEL 8.4 USING THE INSTALLER GUI

Use the following steps to register a newly installed Red Hat Enterprise Linux 8.4 using the RHEL installer GUI.

Prerequisites

- A valid user account on the Red Hat Customer Portal. See the [Create a Red Hat Login page](#) .
If the user account has appropriate entitlements (or the account operates in Simple Content Access mode) they can register using username and password only, without presenting an activation key.
- A valid Activation Key and Organization id

Procedure

1. Authenticate your Red Hat account using the **Account** or **Activation Key** option.

CONNECT TO RED HAT

RED HAT ENTERPRISE LINUX 8.4 INSTALLATION

Done us Help!

Authentication ☒ Account ☐ Activation Key

User name

Password

Purpose ☐ Set System Purpose

Insights ☐ Connect to Red Hat Insights

Options

Not registered.

Register

2. Select the **Set System Purpose** field and from the drop-down menu select the **Role, SLA,** and **Usage for the RHEL 8.4 installation**

CONNECT TO RED HAT

RED HAT ENTERPRISE LINUX 8.4 INSTALLATION

Done us Help!

The system has been properly subscribed Unregister

Method Registered with account

System Purpose Role: Red Hat Enterprise Linux Workstation

SLA:

Usage: Production

Insights Not connected to Red Hat Insights

No subscriptions are attached to the system

At this point, your Red Hat Enterprise Linux 8.4 system has been successfully registered.

CHAPTER 5. 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 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.

5.1. ENABLING OR DISABLING THE SERVICES

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

This section describes the steps for enabling or disabling those services on an already installed operating system:

Prerequisites

- You must have root access to the system.

Procedure

1. To enable a service, use the **enable** option:

```
# systemctl enable service_name
```

Replace *service_name* with the service you want to enable.

You can also enable and start a service in a single command:

```
# systemctl enable --now service_name
```

2. To disable a service, use the **disable** option:

```
# systemctl disable service_name
```

Replace *service_name* with the service you want to disable.



WARNING

You cannot enable a service that has been previously masked. You have to unmask it first:

```
# systemctl unmask service_name
```

5.2. MANAGING SERVICES IN THE RHEL WEB CONSOLE

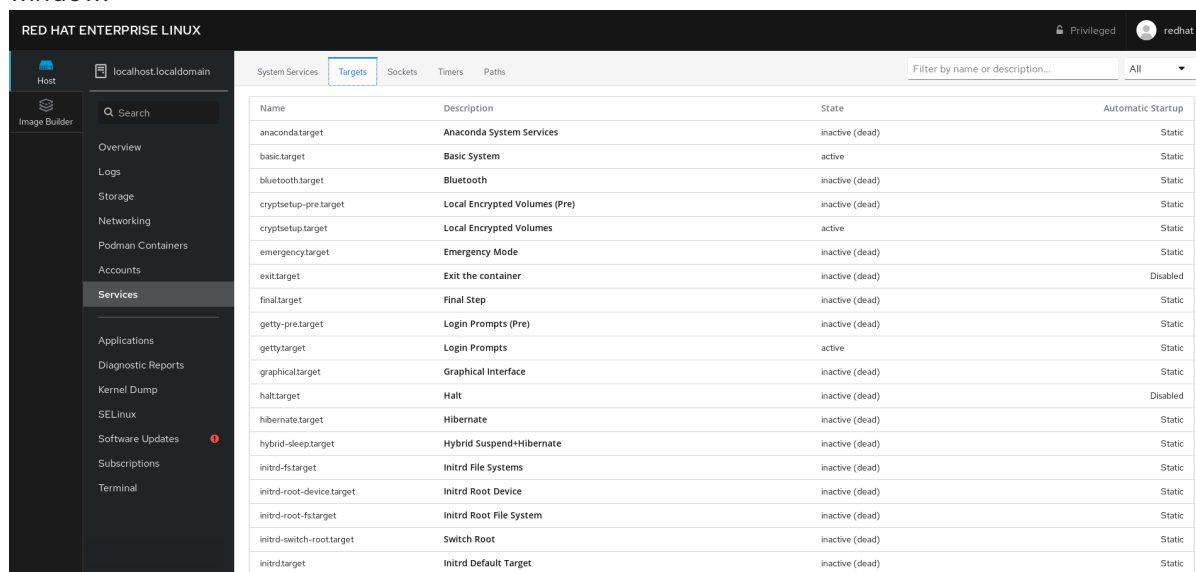
This section describes how you can also enable or disable a service using the web console. You can manage systemd targets, services, sockets, timers, and paths. You can also check the service status, start or stop services, enable or disable them.

Prerequisites

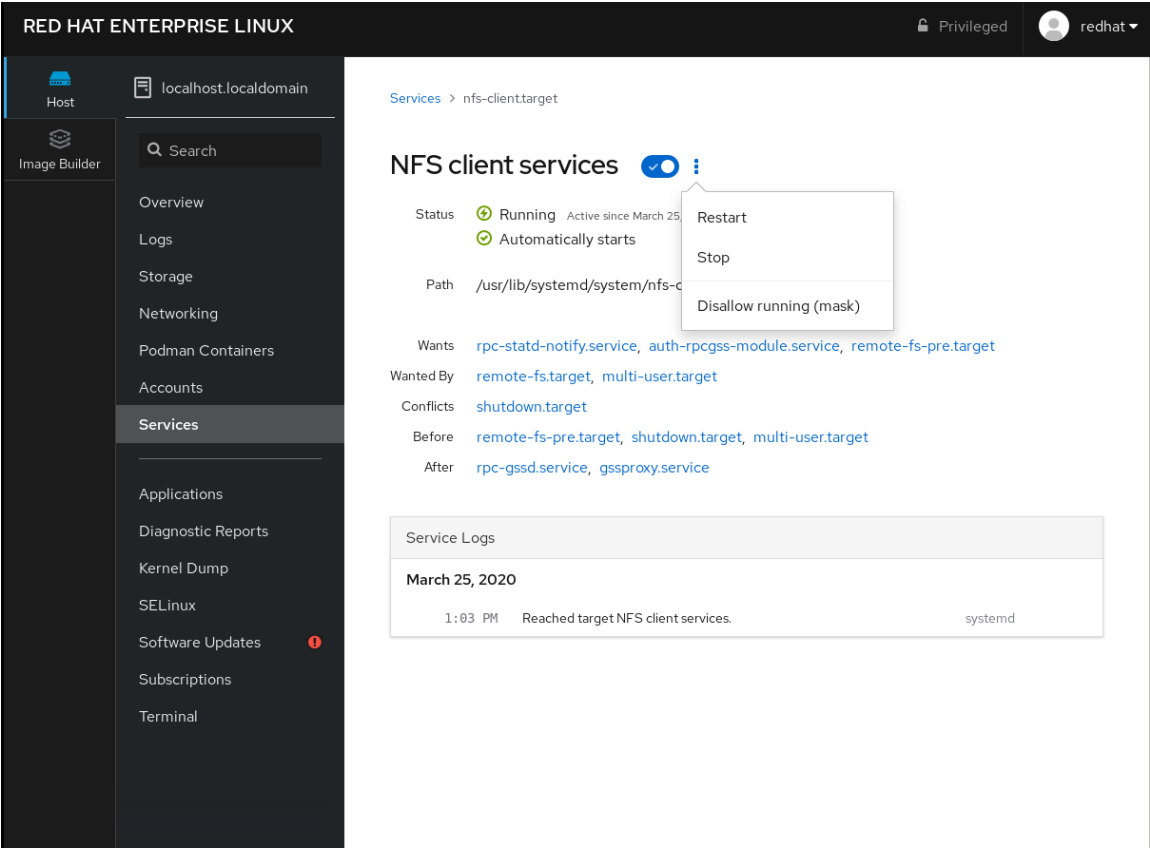
- You must have root access to the system.

Procedure

1. Open <https://localhost:9090/> in a web browser of your preference.
2. Log in to the web console with your root credentials on the system.
3. To display the web console panel, click the **Host** icon, which is in the upper-left corner of the window.



4. On the menu, click **Services**.
You can manage systemd targets, services, sockets, timers, and paths.
5. For example, to manage the service **NFS client services**:
 - a. Click **Targets**.
 - b. Select the service **NFS client services**.
 - c. To enable or disable the service, click the **Toogle** button.
 - d. To stop the service, click the **Stop** button and choose the option **Stop**.



CHAPTER 6. CONFIGURING SYSTEM SECURITY

Computer security is the protection of computer systems and their hardware, software, information, and services from theft, damage, disruption, and misdirection. Ensuring computer security is an essential task, in particular in enterprises that process sensitive data and handle business transactions.

This section covers only the basic security features that you can configure after installation of the operating system.

6.1. ENABLING THE FIREWALLD SERVICE

A firewall is a network security system that monitors and controls incoming and outgoing network traffic according to configured security rules. A firewall typically establishes a barrier between a trusted secure internal network and another outside network.

The **firewalld** service, which provides a firewall in Red Hat Enterprise Linux, is automatically enabled during installation.

To enable the **firewalld** service, follow this procedure.

Procedure

- Display the current status of **firewalld**:

```
$ systemctl status firewalld
● firewalld.service - firewalld - dynamic firewall daemon
   Loaded: loaded (/usr/lib/systemd/system/firewalld.service; disabled; vendor preset:
   enabled)
   Active: inactive (dead)
   ...
```

- If **firewalld** is not enabled and running, switch to the **root** user, and start the **firewalld** service and enable to start it automatically after the system restarts:

```
# systemctl enable --now firewalld
```

Verification steps

- Check that **firewalld** is running and enabled:

```
$ systemctl status firewalld
● firewalld.service - firewalld - dynamic firewall daemon
   Loaded: loaded (/usr/lib/systemd/system/firewalld.service; enabled; vendor preset:
   enabled)
   Active: active (running)
   ...
```

Additional resources

- [Using and configuring firewalls](#)
- **man firewalld(1)**

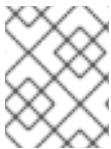
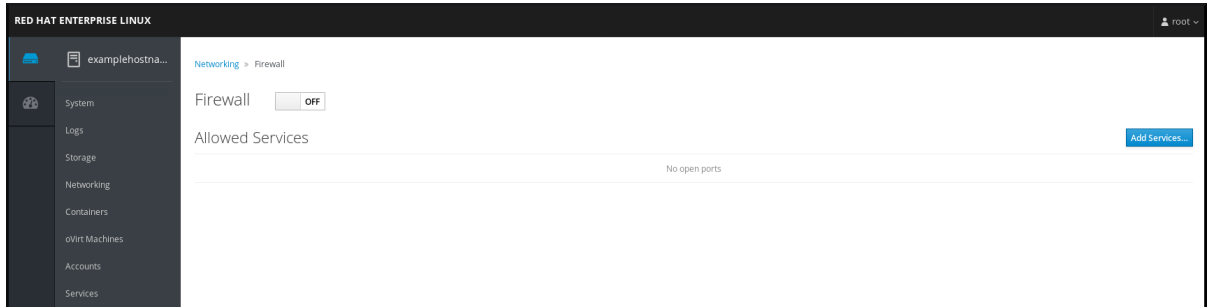
6.2. MANAGING FIREWALL IN THE RHEL 8 WEB CONSOLE

To configure the **firewalld** service in the web console, navigate to **Networking → Firewall**.

By default, the **firewalld** service is enabled.

Procedure

1. To enable or disable **firewalld** in the web console, switch the **Firewall** toggle button.



NOTE

Additionally, you can define more fine-grained access through the firewall to a service using the **Add services...** button.

6.3. MANAGING BASIC SELINUX SETTINGS

Security-Enhanced Linux (SELinux) is an additional layer of system security that determines which processes can access which files, directories, and ports. These permissions are defined in SELinux policies. A policy is a set of rules that guide the SELinux security engine.

SELinux has two possible states:

- Disabled
- Enabled

When SELinux is enabled, it runs in one of the following modes:

- Enabled
 - Enforcing
 - Permissive

In **enforcing mode**, SELinux enforces the loaded policies. SELinux denies access based on SELinux policy rules and enables only the interactions that are explicitly allowed. Enforcing mode is the safest SELinux mode and is the default mode after installation.

In **permissive mode**, SELinux does not enforce the loaded policies. SELinux does not deny access, but reports actions that break the rules to the **/var/log/audit/audit.log** log. Permissive mode is the default mode during installation. Permissive mode is also useful in some specific cases, for example when troubleshooting problems.

Additional resources

- [Using SELinux](#)

6.4. ENSURING THE REQUIRED STATE OF SELINUX

By default, SELinux operates in enforcing mode. However, in specific scenarios, you can set SELinux to permissive mode or even disable it.



IMPORTANT

Red Hat recommends to keep your system in enforcing mode. For debugging purposes, you can set SELinux to permissive mode.

Follow this procedure to change the state and mode of SELinux on your system.

Procedure

1. Display the current SELinux mode:

```
$ getenforce
```

2. To temporarily set SELinux:

- a. To Enforcing mode:

```
# setenforce Enforcing
```

- b. To Permissive mode:

```
# setenforce Permissive
```



NOTE

After reboot, SELinux mode is set to the value specified in the **/etc/selinux/config** configuration file.

3. To set SELinux mode to persist across reboots, modify the **SELINUX** variable in the **/etc/selinux/config** configuration file.

For example, to switch SELinux to enforcing mode:

```
# 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
...
```

**WARNING**

Disabling SELinux reduces your system security. Avoid disabling SELinux using the **SELINUX=disabled** option in the `/etc/selinux/config` file because this can result in memory leaks and race conditions causing kernel panics. Instead, disable SELinux by adding the **selinux=0** parameter to the kernel command line [Changing SELinux modes at boot time](#)

Additional resources

- [Changing SELinux states and modes](#)

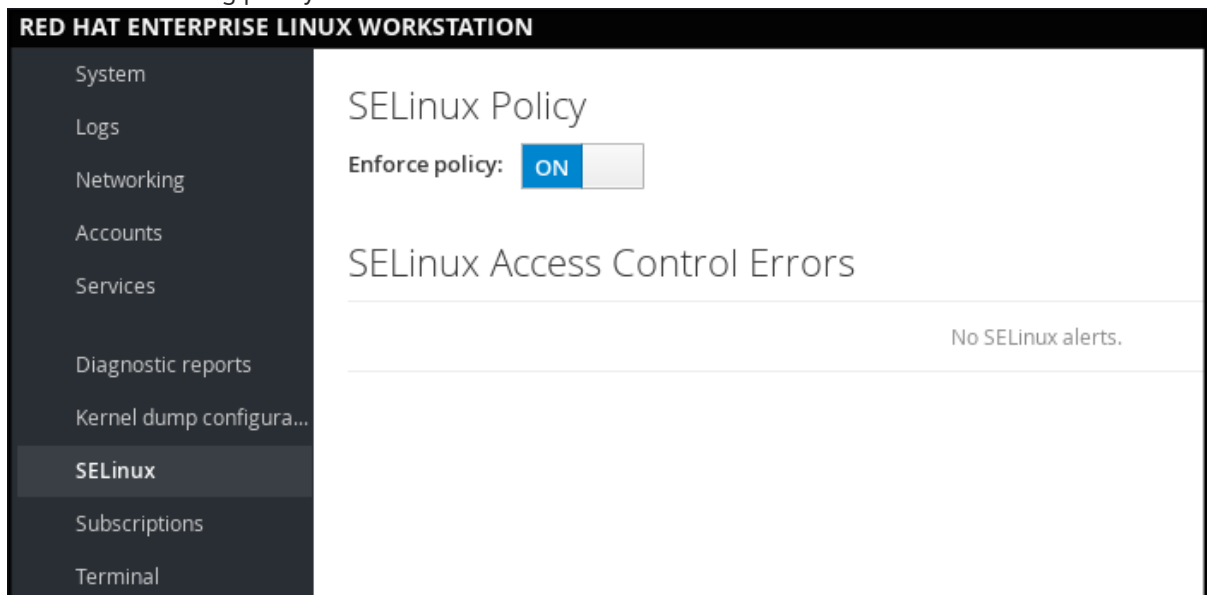
6.5. SWITCHING SELINUX MODES IN THE RHEL 8 WEB CONSOLE

You can set SELinux mode through the RHEL 8 web console in the **SELinux** menu item.

By default, SELinux enforcing policy in the web console is on, and SELinux operates in enforcing mode. By turning it off, you switch SELinux to permissive mode. Note that this selection is automatically reverted on the next boot to the configuration defined in the `/etc/sysconfig/selinux` file.

Procedure

1. In the web console, use the **Enforce policy** toggle button in the SELinux menu item to turn SELinux enforcing policy on or off.

**6.6. ADDITIONAL RESOURCES**

- [Generating SSH key pairs](#)
- [Setting an OpenSSH server for key-based authentication](#)
- [Security hardening](#)
- [Using SELinux](#)

- [Securing networks](#)
- [Deploying the same SELinux configuration on multiple systems](#)

CHAPTER 7. GETTING STARTED WITH 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.

The following are the different types of user accounts:

- **Normal user accounts:**

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

- **System user accounts:**

System user 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. For assigning IDs, see the `/etc/login.defs` file.

- **Group:**

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

7.1. MANAGING ACCOUNTS AND GROUPS USING COMMAND LINE TOOLS

This section describes basic command-line tools to manage user accounts and groups.

- To display user and group IDs:

```
$ id
uid=1000(example.user) gid=1000(example.user) groups=1000(example.user),10(wheel)
context=unconfined_u:unconfined_r:unconfined_t:s0-s0:c0.c1023
```

- To create a new user account:

```
# useradd example.user
```


- To assign a new password to a user account belonging to *example.user*:

```
# passwd example.user
```

- To add a user to a group:

```
# usermod -a -G example.group example.user
```

Additional resources

- **man useradd(8)**, **man passwd(1)**, and **man usermod(8)**

7.2. SYSTEM USER ACCOUNTS MANAGED IN THE WEB CONSOLE

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

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

The RHEL 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.

After logging into the RHEL web console, you can perform the following operations:

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

7.3. ADDING NEW ACCOUNTS USING THE WEB CONSOLE

Use the following steps for adding user accounts to the system and setting administration rights to the accounts through the RHEL web console.

Prerequisites

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

Procedure

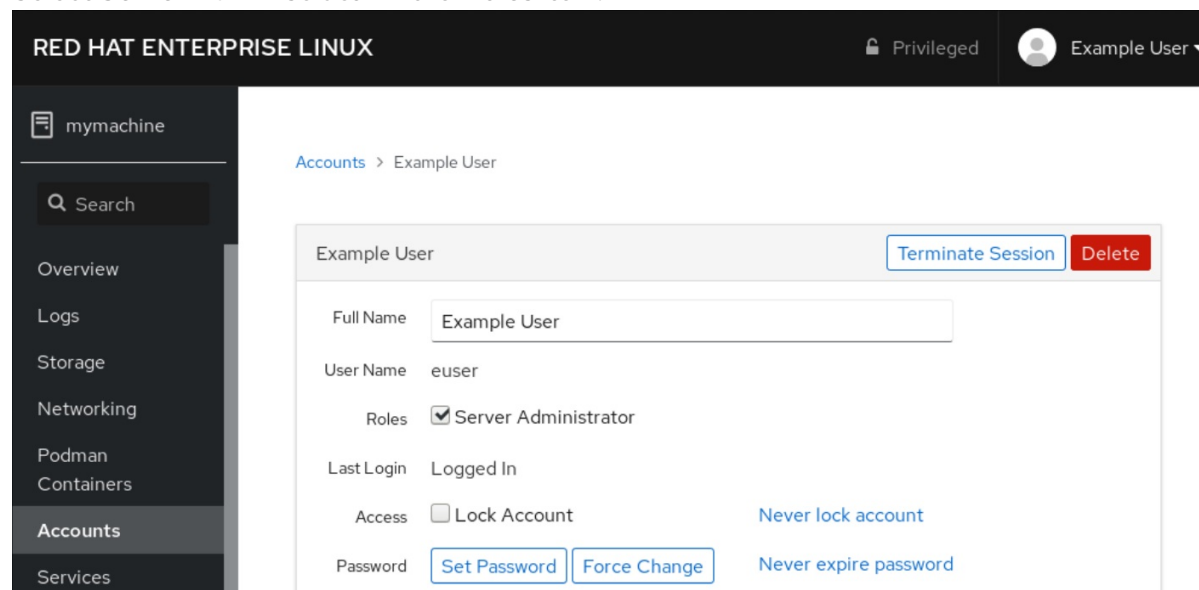
1. Log in to the RHEL web console.
2. Click **Accounts**.
3. Click **Create New Account**.
1. In the **Full Name** field, enter the full name of the user.

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.

2. 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.

1. Click **Create** to save the settings and close the dialog box.
2. Select the newly created account.
3. Select **Server Administrator** in the **Roles** item.



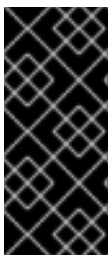
Now you can see the new account in the **Accounts** settings and you can use the credentials to connect to the system.

CHAPTER 8. DUMPING A CRASHED KERNEL FOR LATER ANALYSIS

To analyze why a system crashed, you can use the **kdump** service to save the contents of the system's memory for later analysis. This section provides a brief introduction to **kdump**, and information about configuring **kdump** using the RHEL web console or using the corresponding RHEL system role.

8.1. WHAT IS KDUMP

kdump is a service providing a crash dumping mechanism. The service enables you to save the contents of the system's memory for later analysis. **kdump** uses the **kexec** system call to boot into the second kernel (a *capture kernel*) without rebooting; and then captures the contents of the crashed kernel's memory (a *crash dump* or a *vmcore*) and saves it. The second kernel resides in a reserved part of the system memory.



IMPORTANT

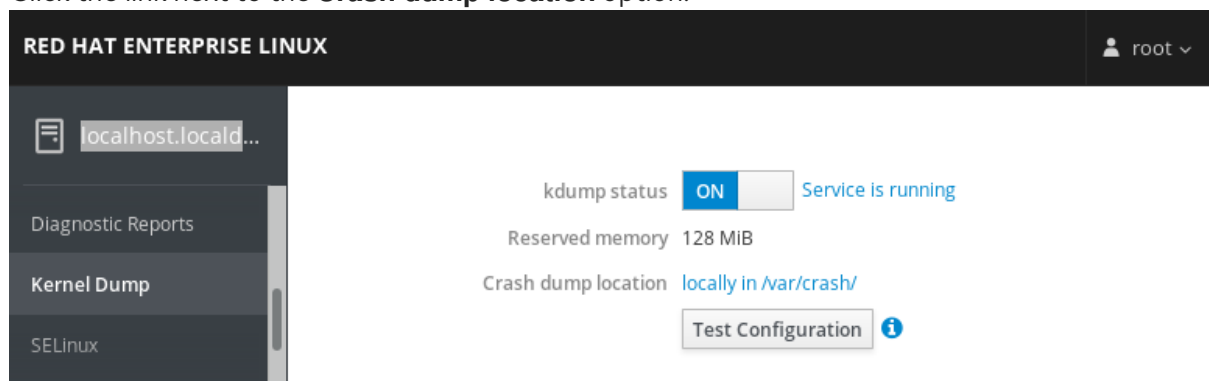
A kernel crash dump can be the only information available in the event of a system failure (a critical bug). Therefore, ensuring that **kdump** is operational is important in mission-critical environments. Red Hat advise that system administrators regularly update and test **kexec-tools** in your normal kernel update cycle. This is especially important when new kernel features are implemented.

8.2. CONFIGURING KDUMP MEMORY USAGE AND TARGET LOCATION IN WEB CONSOLE

The procedure below shows you how to use the **Kernel Dump** tab in the RHEL web console interface to configure the amount of memory that is reserved for the **kdump** kernel. The procedure also describes how to specify the target location of the **vmcore** dump file and how to test your configuration.


Procedure

1. Open the **Kernel Dump** tab and start the **kdump** service.
2. Configure the **kdump** memory usage using the command line.
3. Click the link next to the **Crash dump location** option.



4. Select the **Local Filesystem** option from the drop-down and specify the directory you want to save the dump in.

Crash dump location

Location Local Filesystem 

Directory /var/crash/

Compression ☒ Compress crash dumps to save space

Cancel

Apply

- Alternatively, select the **Remote over SSH** option from the drop-down to send the vmcore to a remote machine using the SSH protocol.
Fill the **Server**, **ssh key**, and **Directory** fields with the remote machine address, ssh key location, and a target directory.
- Another choice is to select the **Remote over NFS** option from the drop-down and fill the **Mount** field to send the vmcore to a remote machine using the NFS protocol.

**NOTE**

Tick the **Compression** check box to reduce the size of the vmcore file.

- Test your configuration by crashing the kernel.

kdump status ON  Service is running

Reserved memory 128 MiB

Crash dump location locally in /var/crash/

Test Configuration

**WARNING**

This step disrupts execution of the kernel and results in a system crash and loss of data.

Additional resources

- [Supported kdump targets](#)
- [Using secure communications between two systems with OpenSSH](#)

8.3. KDUMP USING RHEL SYSTEM ROLES

RHEL System Roles is a collection of Ansible roles and modules that provide a consistent configuration interface to remotely manage multiple RHEL systems. The **kdump** role enables you to set basic kernel dump parameters on multiple systems.



WARNING

The **kdump** role replaces the **kdump** configuration of the managed hosts entirely by replacing the **/etc/kdump.conf** file. Additionally, if the **kdump** role is applied, all previous **kdump** settings are also replaced, even if they are not specified by the role variables, by replacing the **/etc/sysconfig/kdump** file.

The following example playbook shows how to apply the **kdump** system role to set the location of the crash dump files:

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

For a detailed reference on **kdump** role variables, install the **rhel-system-roles** package, and see the **README.md** or **README.html** files in the **/usr/share/doc/rhel-system-roles/kdump** directory.

Additional resources

- [Introduction to RHEL System Roles](#)

8.4. ADDITIONAL RESOURCES

- [Installing kdump](#)
- [Configuring kdump on the command line](#)
- [Configuring kdump in the web console](#)

CHAPTER 9. RECOVERING AND RESTORING A SYSTEM

To recover and restore a system using an existing backup, Red Hat Enterprise Linux provides the Relax-and-Recover (ReaR) utility.

You can use the utility as a disaster recovery solution and also for system migration.

The utility enables you to perform the following tasks:

- Produce a bootable image and restore the system from an existing backup, using the image.
- Replicate the original storage layout.
- Restore user and system files.
- Restore the system to a different hardware.

Additionally, for disaster recovery, you can also integrate certain backup software with ReaR.

Setting up ReaR involves the following high-level steps:

1. Install ReaR.
2. Create rescue system.
3. Modify ReaR configuration file, to add backup method details.
4. Generate backup files.

9.1. SETTING UP REAR

Use the following steps to install the packages for using the Relax-and-Recover (ReaR) utility, create a rescue system, configure and generate a backup.

Prerequisites

- Necessary configurations as per the backup restore plan are ready.
Note that you can use the **NETFS** backup method, a fully-integrated and built-in method with ReaR.

Procedure

1. Install ReaR, the **genisoimage** pre-mastering program, and the **syslinux** package providing a suite of boot loaders:

```
# yum install rear genisoimage syslinux
```

2. Create a rescue system:

```
# rear mkrescue
```

3. Modify the ReaR configuration file in an editor of your choice, for example:

```
# vi /etc/rear/local.conf
```

4. Add the backup setting details to **/etc/rear/local.conf**. For example, in the case of the **NETFS** backup method, add the following lines:

```
BACKUP=NETFS
BACKUP_URL=backup.location
```

Replace *backup.location* by the URL of your backup location.

5. To configure ReaR to keep the previous backup archives when the new ones are created, also add the following line to the configuration file:

```
NETFS_KEEP_OLD_BACKUP_COPY=y
```

6. To make the backups incremental, meaning that only the changed files are backed up on each run, add the following line:

```
BACKUP_TYPE=incremental
```

7. Take a backup as per the restore plan.

CHAPTER 10. TROUBLESHOOTING PROBLEMS USING LOG FILES

Log files contain messages about the system, including the kernel, services, and applications running on it. These contain information that helps troubleshoot issues or monitor system functions. 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.

10.1. SERVICES HANDLING SYSLOG MESSAGES

The following two services handle **syslog** messages:

- The **systemd-journald** daemon
- The **Rsyslog** service

The **systemd-journald** daemon collects messages from various sources and forwards them to **Rsyslog** for further processing. The **systemd-journald** daemon collects messages from the following sources:

- Kernel
- Early stages of the boot process
- Standard and error output of daemons as they start up and run
- **Syslog**

The **Rsyslog** service sorts the **syslog** messages by type and priority and writes them to the files in the **/var/log** directory. The **/var/log** directory persistently stores the log messages.

10.2. SUBDIRECTORIES STORING SYSLOG MESSAGES

The following subdirectories under the **/var/log** directory store **syslog** messages.

- **/var/log/messages** - all **syslog** messages except the following
- **/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

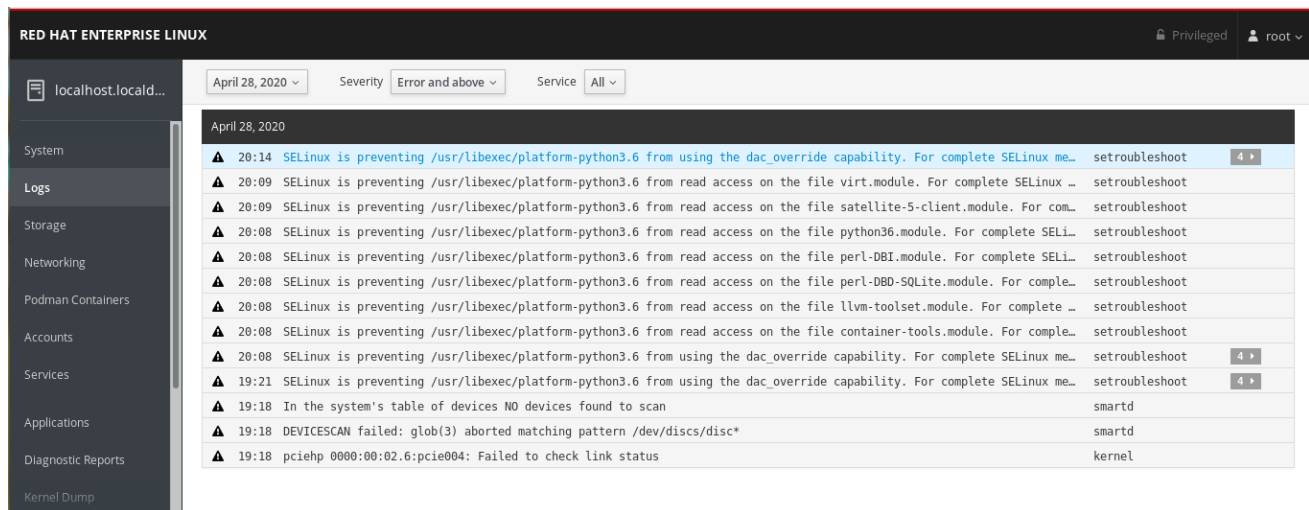
10.3. INSPECTING LOG FILES USING THE WEB CONSOLE

Follow the steps in this procedure to inspect the log files using the web console.

Procedure

1. Log into the Red Hat Enterprise Linux web console.
2. Click **Logs**.

Figure 10.1. Inspecting the log files in the RHEL 8 web console



Additional resources

- [Logging in to the web console](#)

10.4. VIEWING LOGS USING THE COMMAND LINE

The Journal is a component of systemd that helps to view and manage log files. It addresses problems connected with traditional logging, closely integrated with the rest of the system, and supports various logging technologies and access management for the log files.

You can use the **journalctl** command to view messages in the system journal using the command line, for example:

```
$ journalctl -b | grep kvm
May 15 11:31:41 localhost.localdomain kernel: kvm-clock: Using msrs 4b564d01 and 4b564d00
May 15 11:31:41 localhost.localdomain kernel: kvm-clock: cpu 0, msr 76401001, primary cpu clock
...
```

Table 10.1. Viewing system information

Command	Description
journalctl	Shows all collected journal entries.
journalctl FILEPATH	Shows logs related to a specific file. For example, the journalctl /dev/sda command displays logs related to the /dev/sda file system.
journalctl -b	Shows logs for the current boot.
journalctl -k -b -1	Shows kernel logs for the current boot.

Table 10.2. Viewing information on specific services

Command	Description
<code>journalctl -b _SYSTEMD_UNIT=foo</code>	Filters log to see ones matching the "foo" systemd service.
<code>journalctl -b _SYSTEMD_UNIT=foo _PID=number</code>	Combines matches. For example, this command shows logs for systemd-units that match foo and the PID number .
<code>journalctl -b _SYSTEMD_UNIT=foo _PID=number + _SYSTEMD_UNIT=foo1</code>	The separator "+" combines two expressions in a logical OR. For example, this command shows all messages from the foo service process with the PID plus all messages from the foo1 service (from any of its processes).
<code>journalctl -b _SYSTEMD_UNIT=foo _SYSTEMD_UNIT=foo1</code>	This command shows all entries matching either expression, referring to the same field. Here, this command shows logs matching a systemd-unit foo or a systemd-unit foo1 .

Table 10.3. Viewing logs related to specific boots

Command	Description
<code>journalctl --list-boots</code>	Shows a tabular list of boot numbers, their IDs, and the timestamps of the first and last message pertaining to the boot. You can use the ID in the next command to view detailed information.
<code>journalctl --boot=ID _SYSTEMD_UNIT=foo</code>	Shows information about the specified boot ID.

10.5. ADDITIONAL RESOURCES

- `man journalctl(1)`
- [Configuring a remote logging solution](#)

CHAPTER 11. ACCESSING THE RED HAT SUPPORT

This section describes how to effectively troubleshoot your problems using Red Hat support and **sosreport**.

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

11.1. OBTAINING RED HAT SUPPORT THROUGH RED HAT CUSTOMER PORTAL

The following section describes how to use the Red Hat Customer Portal to get help.

Prerequisites

- A valid user account on the Red Hat Customer Portal. See [Create a Red Hat Login](#) .
- An active subscription for the RHEL system.

Procedure

1. Access [Red Hat support](#):
 - a. Open a new support case.
 - b. Initiate a live chat with a Red Hat expert.
 - c. Contact a Red Hat expert by making a call or sending an email.

11.2. TROUBLESHOOTING PROBLEMS USING SOSREPORT

The **sosreport** command collects configuration details, system information and diagnostic information from a Red Hat Enterprise Linux system.

The following section describes how to use the **sosreport** command to produce reports for your support cases.

Prerequisites

- A valid user account on the Red Hat Customer Portal. See [Create a Red Hat Login](#) .
- An active subscription for the RHEL system.
- A support-case number.

Procedure

1. Install the **sos** package:

```
# yum install sos
```

**NOTE**

The default minimal installation of Red Hat Enterprise Linux does not include the **sos** package, which provides the **sosreport** command.

2. Generate a report:

```
# sosreport
```

3. Attach the report to your support case.

See the [How can I attach a file to a Red Hat support case?](#) Red Hat Knowledgebase article for more information.

Note that when attaching the report, you are prompted to enter the number of the relevant support case.

Additional resources

- [What is a sosreport and how to create one in Red Hat Enterprise Linux 4.6 and later?](#)

CHAPTER 12. MANAGING SOFTWARE PACKAGES

12.1. SOFTWARE MANAGEMENT TOOLS IN RED HAT ENTERPRISE LINUX 8

In RHEL 8, software installation is enabled by the new version of the **YUM** tool (**YUM v4**), which is based on the **DNF** technology.



NOTE

Upstream documentation identifies the technology as **DNF** and the tool is referred to as **DNF** in the upstream. As a result, some output returned by the new **YUM** tool in RHEL 8 mentions **DNF**.

Although **YUM v4** used in RHEL 8 is based on **DNF**, it is compatible with **YUM v3** used in RHEL 7. For software installation, the **yum** command and most of its options work the same way in RHEL 8 as they did in RHEL 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 RHEL 7. Packages 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. You can migrate your plug-ins and scripts to the new API provided by **YUM v4** (DNF Python API), which is stable and fully supported. See [DNF API Reference](#) for more information.

12.2. APPLICATION STREAMS

Red Hat Enterprise Linux 8 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 Red Hat Enterprise Linux 8. Each Application Stream has a given life cycle, either the same as RHEL 8 or shorter, more suitable to the particular application. Application Streams with a shorter life cycle are listed in the [Red Hat Enterprise Linux 8 Application Streams Life Cycle](#) page.

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](#).

12.3. SEARCHING FOR SOFTWARE PACKAGES

yum allows you to perform a complete set of operations with software packages.

The following section describes how to use **yum** to:

- Search for packages.
- List packages.
- List repositories.
- Display information about the packages.
- List package groups.
- Specify global expressions in yum input.

12.3.1. Searching packages with yum

- To search for a package, use:

```
# yum search term
```

Replace *term* with a term related to the package.

Note that **yum search** command returns term matches within the name and summary of the packages. This makes the search faster and enables you to search for packages you do not know the name of, but for which you know a related term.

- To include term matches within package descriptions, use:

```
# yum search --all term
```

Replace *term* with a term you want to search for in a package name, summary, or description.

Note that **yum search --all** enables a more exhaustive but slower search.

12.3.2. Listing packages with yum

- To list information on all installed and available packages, use:

```
# yum list --all
```

- To list all packages installed on your system, use:

```
# yum list --installed
```

- To list all packages in all enabled repositories that are available to install, use:

```
# yum list --available
```

Note that you can filter the results by appending global expressions as arguments. See [Section 12.3.6, “Specifying global expressions in yum input”](#) for more details.

12.3.3. Listing repositories with yum

- To list all enabled repositories on your system, use:

```
# yum repolist
```

- To list all disabled repositories on your system, use:

```
# yum repolist --disabled
```

- To list both enabled and disabled repositories, use:

```
# yum repolist --all
```

- To list additional information about the repositories, use:

```
# yum repoinfo
```

Note that you can filter the results by passing the ID or name of repositories as arguments or by appending global expressions. See [Section 12.3.6, “Specifying global expressions in yum input”](#) for more details.

12.3.4. Displaying package information with yum

- To display information about one or more packages, use:

```
# yum info package-name
```

Replace *package-name* with the name of the package.

Note that you can filter the results by appending global expressions as arguments. See [Section 12.3.6, “Specifying global expressions in yum input”](#) for more details.

12.3.5. Listing package groups with yum

- To view the number of installed and available groups, use:

```
# yum group summary
```

- To list all installed and available groups, use:

```
# yum group list
```

Note that you can filter the results by appending command line options for the **yum group list** command (**--hidden**, **--available**). For more available options see the man pages.

- To list mandatory and optional packages contained in a particular group, use:

```
# yum group info group-name
```

Replace *group-name* with the name of the group.

Note that you can filter the results by appending global expressions as arguments. See [Section 12.7.4, “Specifying global expressions in yum input”](#) for more details.

12.3.6. Specifying global expressions in yum input

yum commands allow you to filter the results by appending one or more *glob expressions* as arguments. Global expressions must be escaped when passed as arguments to the **yum** command. To ensure global expressions are passed to **yum** as intended, use one of the following methods:

- Double-quote or single-quote the entire global expression.

```
# yum provides "*/file-name"
```

Replace *file-name* with the name of the file.

- Escape the wildcard characters by preceding them with a backslash (\) character.

```
# yum provides \*/file-name
```

Replace *file-name* with the name of the file.

12.4. INSTALLING SOFTWARE PACKAGES

The following section describes how to use **yum** to:

- Install packages.
- Install a package group.
- Specify a package name in yum input.

12.4.1. Installing packages with yum

- To install a package and all the package dependencies, use:

```
# yum install package-name
```

Replace *package-name* with the name of the package.

- To install multiple packages and their dependencies simultaneously, use:

```
# yum install package-name-1 package-name-2
```

Replace *package-name-1* and *package-name-2* with the names of the packages.

- When installing packages on a *multilib* system (AMD64, Intel 64 machine), you can specify the architecture of the package by appending it to the package name:

```
# yum install package-name.arch
```

Replace *package-name.arch* with the name and architecture of the package.

- If you know the name of the binary you want to install, but not the package name, you can use the path to the binary as an argument:

```
# yum install /usr/sbin/binary-file
```

Replace */usr/sbin/binary-file* with a path to the binary file.

yum searches through the package lists, finds the package which provides */usr/sbin/binary-file*, and prompts you as to whether you want to install it.

- To install a previously-downloaded package from a local directory, use:

```
# yum install /path/
```

Replace */path/* with the path to the package.

Note that you can optimize the package search by explicitly defining how to parse the argument. See [Section 12.4.3, “Specifying a package name in yum input”](#) for more details.

12.4.2. Installing a package group with yum

- To install a package group by a group name, use:

```
# yum group install group-name
```

Or

```
# yum install @group-name
```

Replace *group-name* with the full name of the group or environmental group.

- To install a package group by the groupID, use:

```
# yum group install groupID
```

Replace *groupID* with the ID of the group.

12.4.3. Specifying a package name in yum input

To optimize the installation and removal process, you can append **-n**, **-na**, or **-nevra** suffixes to **yum install** and **yum remove** commands to explicitly define how to parse an argument:

- To install a package using its exact name, use:

```
# yum install-n name
```

Replace *name* with the exact name of the package.

- To install a package using its exact name and architecture, use:

```
# yum install-na name.architecture
```

Replace *name* and *architecture* with the exact name and architecture of the package.

- To install a package using its exact name, epoch, version, release, and architecture, use:

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

Replace *name*, *epoch*, *version*, *release*, and *architecture* with the exact name, epoch, version, release, and architecture of the package.

12.5. UPDATING SOFTWARE PACKAGES

yum allows you to check if your system has any pending updates. You can list packages that need updating and choose to update a single package, multiple packages, or all packages at once. If any of the packages you choose to update have dependencies, they are updated as well.

The following section describes how to use **yum** to:

- Check for updates.
- Update a single package.
- Update a package group.
- Update all packages and their dependencies.
- Apply security updates.
- Automate software updates.

12.5.1. Checking for updates with yum

- To see which packages installed on your system have available updates, use:

```
# yum check-update
```

The output returns the list of packages and their dependencies that have an update available.

12.5.2. Updating a single package with yum

- To update a package, use:

```
# yum update package-name
```

Replace *package-name* with the name of the package.



IMPORTANT

When applying updates to kernel, **yum** always **installs** a new kernel regardless of whether you are using the **yum update** or **yum install** command.

12.5.3. Updating a package group with yum

- To update a package group, use:

```
# yum group update group-name
```

Replace *group-name* with the name of the package group.

12.5.4. Updating all packages and their dependencies with yum

- To update all packages and their dependencies, use:

```
# yum update
```

-

12.5.5. Updating security-related packages with yum

- To upgrade to the latest available packages that have security errata, use:

```
# yum update --security
```

- To upgrade to the last security errata packages, use:

```
# yum update-minimal --security
```

12.5.6. Automating software updates

To check and download package updates automatically and regularly, you can use the **DNF Automatic** tool that is provided by the **dnf-automatic** package.

DNF Automatic is an alternative command-line interface to **yum** that is suited for automatic and regular execution using systemd timers, cron jobs and other such tools.

DNF Automatic synchronizes package metadata as needed and then checks for updates available. After, the tool can perform one of the following actions depending on how you configure it:

- Exit
- Download updated packages
- Download and apply the updates

The outcome of the operation is then reported by a selected mechanism, such as the standard output or email.

12.5.6.1. Installing DNF Automatic

The following procedure describes how to install the **DNF Automatic** tool.

Procedure

- To install the **dnf-automatic** package, use:

```
# yum install dnf-automatic
```

Verification steps

- To verify the successful installation, confirm the presence of the **dnf-automatic** package by running the following command:

```
# rpm -qi dnf-automatic
```

12.5.6.2. DNF Automatic configuration file

By default, **DNF Automatic** uses **/etc/dnf/automatic.conf** as its configuration file to define its behavior.

The configuration file is separated into the following topical sections:

- **[commands]** section
Sets the mode of operation of **DNF Automatic**.
- **[emitters]** section
Defines how the results of **DNF Automatic** are reported.
- **[command_email]** section
Provides the email emitter configuration for an external command used to send email.
- **[email]** section
Provides the email emitter configuration.
- **[base]** section
Overrides settings from the main configuration file of yum.

With the default settings of the `/etc/dnf/automatic.conf` file, **DNF Automatic** checks for available updates, downloads them, and reports the results as standard output.



WARNING

Settings of the operation mode from the **[commands]** section are overridden by settings used by a systemd timer unit for all timer units except **dnf-automatic.timer**.

Additional resources

- For more details on particular sections, see [DNF Automatic documentation](#).
- For more details on systemd timer units, see the **man dnf-automatic** manual pages.
- For the overview of the systemd timer units included in the **dnf-automatic package**, see [Section 2.5.6.4 Overview of the systemd timer units included in the dnf-automatic package](#)

12.5.6.3. Enabling DNF Automatic

To run **DNF Automatic**, you always need to enable and start a specific systemd timer unit. You can use one of the timer units provided in the **dnf-automatic** package, or you can write your own timer unit depending on your needs.

The following section describes how to enable **DNF Automatic**.

Prerequisites

- You specified the behavior of DNF Automatic by modifying the `/etc/dnf/automatic.conf` configuration file.

For more information on **DNF Automatic** configuration file, see [Section 2.5.6.2, “DNF Automatic configuration file”](#).

Procedure

- Select, enable and start a systemd timer unit that fits your needs:

```
# systemctl enable --now <unit>
```

where **<unit>** is one of the following timers:

- **dnf-automatic-download.timer**
- **dnf-automatic-install.timer**
- **dnf-automatic-notifyonly.timer**
- **dnf-automatic.timer**

For **downloading** available updates, use:

```
# systemctl enable dnf-automatic-download.timer
```

```
# systemctl start dnf-automatic-download.timer
```

For **downloading and installing** available updates, use:

```
# systemctl enable dnf-automatic-install.timer
```

```
# systemctl start dnf-automatic-install.timer
```

For **reporting** about available updates, use:

```
# systemctl enable dnf-automatic-notifyonly.timer
```

```
# systemctl start dnf-automatic-notifyonly.timer
```

Optionally, you can use:

```
# systemctl enable dnf-automatic.timer
```

```
# systemctl start dnf-automatic.timer
```

In terms of downloading and applying updates, this timer unit behaves according to settings in the **/etc/dnf/automatic.conf** configuration file. The default behavior is similar to **dnf-automatic-download.timer**: it downloads the updated packages, but it does not install them.



NOTE

Alternatively, you can also run **DNF Automatic** by executing the **/usr/bin/dnf-automatic** file directly from the command line or from a custom script.

Verification steps

- To verify that the timer is enabled, run the following command:

```
# systemctl status <systemd timer unit>
```

Additional resources

- For more information on the `dnf-automatic` timers, see the **man `dnf-automatic`** manual pages.
- For the overview of the systemd timer units included in the **`dnf-automatic`** package, see Section [2.5.6.4 Overview of the systemd timer units included in the `dnf-automatic` package](#)

12.5.6.4. Overview of the systemd timer units included in the `dnf-automatic` package

The systemd timer units take precedence and override the settings in the `/etc/dnf/automatic.conf` configuration file concerning downloading and applying updates.

For example if you set:

`download_updates = yes`

in the `/etc/dnf/automatic.conf` configuration file, but you have activated the **`dnf-automatic-notifyonly.timer`** unit, the packages will not be downloaded.

The **`dnf-automatic`** package includes the following systemd timer units:

Timer unit	Function	Overrides settings in the <code>/etc/dnf/automatic.conf</code> file?
<code>dnf-automatic-download.timer</code>	Downloads packages to cache and makes them available for updating. Note: This timer unit does not install the updated packages. To perform the installation, you have to execute the <code>dnf update</code> command.	Yes
<code>dnf-automatic-install.timer</code>	Downloads and installs updated packages.	Yes
<code>dnf-automatic-notifyonly.timer</code>	Downloads only repository data to keep repository cache up-to-date and notifies you about available updates. Note: This timer unit does not download or install the updated packages	Yes

Timer unit	Function	Overrides settings in the <code>/etc/dnf/automatic.conf</code> file?
dnf-automatic.timer	<p>The behavior of this timer concerning downloading and applying updates is specified by the settings in the <code>/etc/dnf/automatic.conf</code> configuration file.</p> <p>Default behavior is the same as for the dnf-automatic-download.timer unit: it only downloads packages, but does not install them.</p>	No

Additional resources

- For more information on the **dnf-automatic** timers, see the **man dnf-automatic** manual pages.
- For more information on the **`/etc/dnf/automatic.conf`** configuration file, see Section [2.5.6.2. DNF Automatic configuration file](#)

12.6. UNINSTALLING SOFTWARE PACKAGES

The following section describes how to use **yum** to:

- Remove packages.
- Remove a package group.
- Specify a package name in yum input.

12.6.1. Removing packages with yum

- To remove a particular package and all dependent packages, use:

```
# yum remove package-name
```

Replace *package-name* with the name of the package.

- To remove multiple packages and their dependencies simultaneously, use:

```
# yum remove package-name-1 package-name-2
```

Replace *package-name-1* and *package-name-2* with the names of the packages.



NOTE

yum is not able to remove a package without removing depending packages.

Note that you can optimize the package search by explicitly defining how to parse the argument. See [Section 12.6.3, “Specifying a package name in yum input”](#) for more details.

12.6.2. Removing a package group with yum

- To remove a package group by the group name, use:

```
# yum group remove group-name
```

Or

```
# yum remove @group-name
```

Replace *group-name* with the full name of the group.

- To remove a package group by the groupID, use:

```
# yum group remove groupID
```

Replace *groupID* with the ID of the group.

12.6.3. Specifying a package name in yum input

To optimize the installation and removal process, you can append **-n**, **-na**, or **-nevra** suffixes to **yum install** and **yum remove** commands to explicitly define how to parse an argument:

- To install a package using its exact name, use:

```
# yum install-n name
```

Replace *name* with the exact name of the package.

- To install a package using its exact name and architecture, use:

```
# yum install-na name.architecture
```

Replace *name* and *architecture* with the exact name and architecture of the package.

- To install a package using its exact name, epoch, version, release, and architecture, use:

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

Replace *name*, *epoch*, *version*, *release*, and *architecture* with the exact name, epoch, version, release, and architecture of the package.

12.7. MANAGING SOFTWARE PACKAGE GROUPS

A package group is a collection of packages that serve a common purpose (**System Tools**, **Sound and Video**). Installing a package group pulls a set of dependent packages, which saves time considerably.

The following section describes how to use **yum** to:

- List package groups.

- Install a package group.
- Remove a package group.
- Specify global expressions in yum input.

12.7.1. Listing package groups with yum

- To view the number of installed and available groups, use:

```
# yum group summary
```

- To list all installed and available groups, use:

```
# yum group list
```

Note that you can filter the results by appending command line options for the **yum group list** command (**--hidden**, **--available**). For more available options see the man pages.

- To list mandatory and optional packages contained in a particular group, use:

```
# yum group info group-name
```

Replace *group-name* with the name of the group.

Note that you can filter the results by appending global expressions as arguments. See [Section 12.7.4](#), “[Specifying global expressions in yum input](#)” for more details.

12.7.2. Installing a package group with yum

- To install a package group by a group name, use:

```
# yum group install group-name
```

Or

```
# yum install @group-name
```

Replace *group-name* with the full name of the group or environmental group.

- To install a package group by the groupID, use:

```
# yum group install groupID
```

Replace *groupID* with the ID of the group.

12.7.3. Removing a package group with yum

- To remove a package group by the group name, use:

```
# yum group remove group-name
```

Or

```
# yum remove @group-name
```

Replace *group-name* with the full name of the group.

- To remove a package group by the groupID, use:

```
# yum group remove groupID
```

Replace *groupID* with the ID of the group.

12.7.4. Specifying global expressions in yum input

yum commands allow you to filter the results by appending one or more *glob expressions* as arguments. Global expressions must be escaped when passed as arguments to the **yum** command. To ensure global expressions are passed to **yum** as intended, use one of the following methods:

- Double-quote or single-quote the entire global expression.

```
# yum provides "*/file-name"
```

Replace *file-name* with the name of the file.

- Escape the wildcard characters by preceding them with a backslash (\) character.

```
# yum provides \*/file-name
```

Replace *file-name* with the name of the file.

12.8. HANDLING PACKAGE MANAGEMENT HISTORY

The **yum history** command allows you to review information about the timeline of **yum** transactions, 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. **yum history** command can also be used to undo or redo the transactions.

The following section describes how to use **yum** to:

- List transactions.
- Revert transactions.
- Repeat transactions.
- Specify global expressions in yum input.

12.8.1. Listing transactions with yum

- To display a list of all the latest **yum** transactions, use:

```
# yum history
```

- To display a list of all the latest operations for a selected package, use:

```
# yum history list package-name
```

■

Replace *package-name* with the name of the package. You can filter the command output by appending global expressions. See [Section 12.8.4, “Specifying global expressions in yum input”](#) for more details.

- To examine a particular transaction, use:

```
# yum history info transactionID
```

Replace *transactionID* with the ID of the transaction.

12.8.2. Reverting transactions with yum

- To revert a particular transaction, use:

```
# yum history undo transactionID
```

Replace *transactionID* with the ID of the transaction.

- To revert the last transaction, use:

```
# yum history undo last
```

Note that the **yum history undo** command only reverts the steps that were performed during the transaction. If the transaction installed a new package, the **yum history undo** command uninstalls it. If the transaction uninstalled a package, the **yum history undo** command reinstalls it. **yum history undo** also attempts to downgrade all updated packages to their previous versions, if the older packages are still available.

12.8.3. Repeating transactions with yum

- To repeat a particular transaction, use:

```
# yum history redo transactionID
```

Replace *transactionID* with the ID of the transaction.

- To repeat the last transaction, use:

```
# yum history redo last
```

Note that the **yum history redo** command only repeats the steps that were performed during the transaction.

12.8.4. Specifying global expressions in yum input

yum commands allow you to filter the results by appending one or more *glob expressions* as arguments. Global expressions must be escaped when passed as arguments to the **yum** command. To ensure global expressions are passed to **yum** as intended, use one of the following methods:

- Double-quote or single-quote the entire global expression.

```
# yum provides "*/file-name"
```

Replace *file-name* with the name of the file.

- Escape the wildcard characters by preceding them with a backslash (\) character.

```
# yum provides \*/file-name
```

Replace *file-name* with the name of the file.

12.9. MANAGING SOFTWARE REPOSITORIES

The configuration information for **yum** and related utilities are stored in the `/etc/yum.conf` file. This file contains one or more **[repository]** sections, which allow you to set repository-specific options.

It is recommended to define individual repositories in new or existing **.repo** files in the `/etc/yum.repos.d/` directory.

Note that the values you define in individual **[repository]** sections of the `/etc/yum.conf` file override values set in the **[main]** section.

The following section describes how to:

- Set **[repository]** options.
- Add a **yum** repository.
- Enable a **yum** repository.
- Disable a **yum** repository.

12.9.1. Setting yum repository options

The `/etc/yum.conf` configuration file contains the **[repository]** sections, where *repository* is a unique repository ID. The **[repository]** sections allows you to define individual **yum** repositories.



NOTE

Do not give custom repositories names used by the Red Hat repositories to avoid conflicts.

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

12.9.2. Adding a yum repository

To define a new repository, you can:

- Add a **[repository]** section to the `/etc/yum.conf` file.
- Add a **[repository]** section to a **.repo** file in the `/etc/yum.repos.d/` directory. **yum** repositories commonly provide their own **.repo** file.

**NOTE**

It is recommended to define your repositories in a **.repo** file instead of **/etc/yum.conf** as all files with the **.repo** file extension in this directory are read by **yum**.

- To add a repository to your system and enable it, use:

```
# yum-config-manager --add-repo repository_URL
```

Replace *repository_url* with URL pointing to the repository.

**WARNING**

Obtaining and installing software packages from unverified or untrusted sources other than Red Hat certificate-based **Content Delivery Network (CDN)** constitutes a potential security risk, and could lead to security, stability, compatibility, and maintainability issues.

12.9.3. Enabling a yum repository

- To enable a repository, use:

```
# yum-config-manager --enable repositoryID
```

Replace *repositoryID* with the unique repository ID.

To list available repository IDs, see [Section 12.3.2, “Listing packages with yum”](#).

12.9.4. Disabling a yum repository

- To disable a yum repository, use:

```
# yum-config-manager --disable repositoryID
```

Replace *repositoryID* with the unique repository ID.

To list available repository IDs, see [Section 12.3.2, “Listing packages with yum”](#).

12.10. CONFIGURING YUM

The configuration information for **yum** and related utilities are stored in the **/etc/yum.conf** file. This file contains one mandatory **[main]** section, which enables you to set **yum** options that have global effect.

The following section describes how to:

- View the current **yum** configurations.
- Set **yum** **[main]** options.

- Use **yum** plug-ins.

12.10.1. Viewing the current yum configurations

- To display the current values of global yum options specified in the **[main]** section of the **/etc/yum.conf** file, use:

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

12.10.2. Setting yum main options

The **/etc/yum.conf** configuration file contains one **[main]** section. The key-value pairs in this section affect how **yum** operates and treats repositories.

You can add 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.

12.10.3. Using yum plug-ins

yum provides plug-ins that extend and enhance its operations. Certain plug-ins are installed by default.

The following section describes how to enable, configure, and disable **yum** plug-ins.

12.10.3.1. Managing yum plug-ins

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.

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

12.10.3.2. Enabling yum plug-ins

- To enable all yum plug-ins:
 1. Ensure a line beginning with **plugins=** is present in the **[main]** section of the **/etc/yum.conf** file.
 2. Set the value of **plugins=** to **1**.

```
plugins=1
```

12.10.3.3. Disabling yum plug-ins

- To disable all yum plug-ins:
 1. Ensure a line beginning with **plugins=** is present in the **[main]** section of the **/etc/yum.conf** file.
 2. Set the value of **plugins=** to **0**.

```
plugins=0
```



IMPORTANT

Disabling all plug-ins is **not** advised. 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 advisable only when diagnosing a potential problem with **yum**.

- To disable all yum plug-ins for a particular command, append **--noplugins** option to the command.

```
# yum --noplugins update
```

- To disable certain yum plug-ins for a single command, append **--disableplugin=*plugin-name*** option to the command.

```
# yum update --disableplugin=plugin-name
```

Replace *plugin-name* with the name of the plug-in.

CHAPTER 13. 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 13.1. systemd unit files locations

Directory	Description
/usr/lib/systemd/system/	systemd unit files distributed with installed RPM packages.
/run/systemd/system/	systemd unit files created at run time. This directory takes precedence over the directory with installed service unit files.
/etc/systemd/system/	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.

The units encapsulate information about:

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

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=pass:_required value_
```

13.1. SYSTEMD UNIT TYPES

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

Table 13.2. Available systemd unit types

Unit Type	File Extension	Description
Service unit	.service	A system service.

Unit Type	File Extension	Description
Target unit	.target	A group of systemd units.
Automount unit	.automount	A file system automount point.
Device unit	.device	A device file recognized by the kernel.
Mount unit	.mount	A file system mount point.
Path unit	.path	A file or directory in a file system.
Scope unit	.scope	An externally created process.
Slice unit	.slice	A group of hierarchically organized units that manage system processes.
Socket unit	.socket	An inter-process communication socket.
Swap unit	.swap	A swap device or a swap file.
Timer unit	.timer	A systemd timer.

13.2. SYSTEMD 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.

13.3. 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 [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.
- The **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 [Changing the timeout limit](#).

13.4. ADDITIONAL RESOURCES (OR NEXT STEPS)

- [Migration Planning Guide](#) for Red Hat Enterprise Linux 7

CHAPTER 14. MANAGING SYSTEM SERVICES WITH SYSTEMCTL

The **systemctl** utility helps managing system services. You can use the **systemctl** utility to perform different tasks related to different services such as starting, stopping, restarting, enabling, and disabling services, listing services, and displaying system services statuses.

This section describes how to manage system services with the **systemctl** utility.

14.1. SERVICE UNIT MANAGEMENT WITH SYSTEMCTL

The **service units** help to control the state of services and daemons in your system.

Service units end with the **.service** file extension, for example **nfs-server.service**. However, while using service file names in commands, you can omit the file extension. The **systemctl** utility assumes the argument is a service unit. For example, to stop the **nfs-server.service**, enter the following command:

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

Additionally, some service units have **alias names**. Aliases can be shorter than units, and you can use them instead of the actual unit names.

To find all aliases that can be used for a particular unit, use:

```
# systemctl show nfs-server.service -p Names
```

Additional resources

- [Comparison of a service utility with systemctl](#)
- [Listing system services](#)
- [Displaying system service status](#)
- [Starting a system service](#)
- [Restarting a system service](#)
- [Enabling a system service](#)
- [Disabling a system service](#)
- [Stopping a system service](#)

14.2. COMPARISON OF A SERVICE UTILITY WITH SYSTEMCTL

This section shows a comparison between a service utility and the usage of **systemctl** command.

Table 14.1. Comparison of the service utility with systemctl

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

14.3. LISTING SYSTEM SERVICES

You can list all currently loaded service units and the status of all available service units.

Procedure

- To list all currently loaded service units, enter:

```
$ 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-d.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'
```

By default, the **systemctl list-units** command displays only active units. For each service unit file, the command displays:

- **UNIT**: its full name
 - **LOAD**: information whether the unit file has been loaded
 - **ACTIVE\ SUB**: its high-level and low-level unit file activation state
 - **DESCRIPTION**: a short description
- To list **all loaded units regardless of their state** enter the following command with the **--all** or **-a** command line option:

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

- To list the status (**enabled / disabled**) of all available service units, enter:

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

208 unit files listed.
```

For each service unit, this command displays:

- **UNIT FILE**: its full name
- **STATE**: information whether the service unit is enabled or disabled

Additional resources

- [Displaying system service status](#)

14.4. DISPLAYING SYSTEM SERVICE STATUS

You can inspect any service unit to get its detailed information and verify the state of the service whether it is enabled or running. You can also view services that are ordered to start after or before a particular service unit.

Procedure

- To display detailed information about a service unit that corresponds to a system service, enter:

```
$ 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 [Available service unit information](#), if it is executed by the **root** user, and the most recent log entries.

Table 14.2. Available service unit information

Field	Description
Loaded	Information whether the service unit has been loaded, the absolute path to the unit file, and a note whether the unit is enabled.
Active	Information whether the service unit is running followed by a time stamp.
Main PID	The PID of the corresponding system service followed by its name.
Status	Additional information about the corresponding system service.
Process	Additional information about related processes.
CGroup	Additional information about related Control Groups (cgroups).

Example 14.1. 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
            └─1037 /usr/libexec/gdm-simple-slave --display-id /org/gno...
              └─1047 /usr/bin/Xorg :0 -background none -verbose -auth /r...

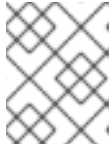
Oct 17 17:31:23 localhost systemd[1]: Started GNOME Display Manager.
```

- To only verify that a particular service unit is running, enter:

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

- To determine whether a particular service unit is enabled, enter:

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

**NOTE**

Both **systemctl is-active** and **systemctl is-enabled** return an exit status of **0** if the specified service unit is running or enabled.

- To determine what services are ordered to start before the specified service unit, enter:

```
# systemctl list-dependencies --after <name>.service
```

Replace *<name>* with the name of the service in the command. For example, to view the list of services ordered to start before **gdm**, enter:

```
# systemctl list-dependencies --after gdm.service
gdm.service
├─dbus.socket
├─getty@tty1.service
├─livesys.service
├─plymouth-quit.service
├─system.slice
├─systemd-journald.socket
├─systemd-user-sessions.service
└─basic.target
[output truncated]
```

- To determine what services are ordered to start after the specified service unit, enter:

```
# systemctl list-dependencies --before <name>.service
```

Replace *<name>* with the name of the service in the command. For example, to view the list of services ordered to start after **gdm**, enter:

```
# 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
```

Additional resources

- [Listing system services](#)

14.5. POSITIVE AND NEGATIVE SERVICE DEPENDENCIES

In **systemd**, positive and negative dependencies between services exist. Starting a 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, without explicit notification to the user. This means that 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 attempt to start the **sendmail** service, **systemd** first automatically stops **postfix**, because these two services are conflicting and cannot run on the same port.

Additional resources

- [Starting a system service](#)

14.6. STARTING A SYSTEM SERVICE

You can start system service in the current session using the **start** command. You must have a **root** access as starting a service may affect the state of the operating system.

Procedure

- To start a selected service unit corresponding to a system service, type the following command as **root**:

```
# systemctl start <name>.service
```

Replace *<name>* with the name of the service unit you want to start (for example, **httpd.service**).

Example 14.2. Starting httpd.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, enter the following command as **root**:

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

Additional resources

- [Positive and negative service dependencies](#)
- [Enabling a system service](#)
- [Displaying system service status](#)

14.7. STOPPING A SYSTEM SERVICE

You can stop system service in the current session using the **stop** command. You must have a **root** access as stopping a service may affect the state of the operating system.

Procedure

- To stop the service unit corresponding to a system service, enter the following command as **root**:

```
# systemctl stop <name>.service
```

Replace *<name>* with the name of the service unit you want to stop (for example, **bluetooth**).

Example 14.3. Stopping bluetoothd.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, enter the following command as **root**:

```
# systemctl stop bluetooth.service
```

Additional resources

- [Disabling a system service](#)
- [Displaying system service status](#)

14.8. RESTARTING A SYSTEM SERVICE

You can restart system service in the current session using the **restart** command. You must have a **root** access as restarting a service may affect the state of the operating system.

This procedure describes how to:

- Stop the selected service unit in the current session and immediately start it again
- Restart a service unit only if the corresponding service is already running
- Reload configuration of a system service without interrupting its execution

Procedure

- To restart a service unit corresponding to a system service, enter the following command as **root**:

```
# systemctl restart <name>.service
```

Replace *<name>* with the name of the service unit you want to restart (for example, **httpd**).



NOTE

If the selected service unit is not running, this command starts it too.

- Alternatively, to restart a service unit only if the corresponding service is already running, enter the following command as **root**:

```
# systemctl try-restart <name>.service
```

- To reload the configuration without interrupting service execution, enter the following command as **root**:

■

```
# systemctl reload <name>.service
```



NOTE

System services that do not support this feature, ignore this command. To restart such services, use the **reload-or-restart** and **reload-or-try-restart** commands instead.

Example 14.4. Reloading httpd.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, enter the following as **root**:

```
# systemctl reload httpd.service
```

Additional resources

- [Displaying system service status](#)

14.9. ENABLING A SYSTEM SERVICE

You can configure service to start automatically at the system booting time. The **enable** 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 sub-directories. However, it does not rewrite links that already exist.

Procedure

- To configure a service unit that corresponds to a system service to be automatically started at boot time, enter the following command as **root**:

```
# systemctl enable <name>.service
```

Replace *<name>* with the name of the service unit you want to enable (for example, **httpd**).

- If you want to ensure that the symbolic links are re-created, enter the following command as **root**:

```
# systemctl reenabale <name>.service
```

This command disables the selected service unit and immediately enables it again.

Example 14.5. Enabling httpd.service

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

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

Additional resources

- [Displaying system service status](#)
- [Starting a system service](#)

14.10. DISABLING A SYSTEM SERVICE

You can prevent a service unit from starting automatically at boot time. The **disable** command reads the **[Install]** section of the selected service unit and removes appropriate symbolic links to the **/usr/lib/systemd/system/*name.service*** file from the **/etc/systemd/system/** directory and its sub-directories.

Procedure

- To configure a service unit that corresponds to a system service not to start automatically at boot time, enter the following command as **root**:

```
# systemctl disable <name>.service
```

Replace *<name>* with the name of the service unit you want to disable (for example, **bluetooth**).

Example 14.6. Disabling bluetoothd.service

The service unit for the **bluetoothd** daemon is named **bluetooth.service**. To prevent this service unit from starting at boot time, enter the following command as a **root**:

```
# systemctl disable bluetooth.service
Removed symlink /etc/systemd/system/bluetooth.target.wants/bluetooth.service.
Removed symlink /etc/systemd/system/dbus-org.bluez.service.
```

- To mask any service unit and prevent it from being started manually or by another service, enter the following command as **root**:

```
# systemctl mask <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, enter:

```
# systemctl unmask <name>.service
```

Additional resources

- [Displaying system service status](#)

- [Stopping a system service](#)

CHAPTER 15. WORKING WITH SYSTEMD TARGETS

systemd targets are represented by target units. Target units file ends 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.

This section includes procedures to implement while working with **systemd** targets.

15.1. DIFFERENCE BETWEEN SYSV RUNLEVELS AND SYSTEMD TARGETS

The previous versions of Red Hat Enterprise Linux were distributed with SysV init or Upstart, and 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.

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. For compatibility reasons, it also provides aliases for these targets that directly maps to the SysV runlevels.

The following table provides a complete list of SysV runlevels and their corresponding systemd targets:

Table 15.1. Comparison of SysV runlevels with systemd targets

Runlevel	Target Units	Description
0	runlevel0.target, poweroff.target	Shut down and power off the system.
1	runlevel1.target, rescue.target	Set up a rescue shell.
2	runlevel2.target, multi-user.target	Set up a non-graphical multi-user system.
3	runlevel3.target, multi-user.target	Set up a non-graphical multi-user system.
4	runlevel4.target, multi-user.target	Set up a non-graphical multi-user system.
5	runlevel5.target, graphical.target	Set up a graphical multi-user system.
6	runlevel6.target, reboot.target	Shut down and reboot the system.

The following table compares the SysV init commands with `systemctl`. Use the `systemctl` utility to view, change, or configure `systemd` targets:



IMPORTANT

The **`runlevel`** and **`telinit`** commands are still available in the system and work as expected, but are only included for compatibility reasons and should be avoided.

Table 15.2. Comparison of SysV init commands with `systemctl`

Old Command	New Command	Description
<code>runlevel</code>	<code>systemctl list-units --type target</code>	Lists currently loaded target units.
<code>telinit <i>runlevel</i></code>	<code>systemctl isolate <i>name.target</i></code>	Changes the current target.

Additional resources

- man **`sysv init`**
- man **`upstart init`**
- man **`systemctl`**

15.2. VIEWING THE DEFAULT TARGET

The default target unit is represented by the `/etc/systemd/system/default.target` file.

Procedure

- To determine which target unit is used by default:

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

- To determine the default target using the symbolic link:

```
$ ls -l /usr/lib/systemd/system/default.target
```

= Viewing the target units

By default, the **`systemctl list-units`** command displays only active units.

Procedure

- To list all loaded units regardless of their state:

```
$ systemctl list-units --type target --all
```

- To list all currently loaded target units:

```
$ 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.

15.2.1. Changing the default target

The default target unit is represented by the **/etc/systemd/system/default.target** file. The following procedure describes how to change the default target by using the `systemctl` command:

Procedure

1. To determine the default target unit:

```
# systemctl get-default
```

2. To configure the system to use a different target unit by default:

```
# 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
```

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. Replace *multi-user* with the name of the target unit you want to use by default.

3. Reboot

```
# reboot
```

15.2.2. Changing the default target using symbolic link

The following procedure describes how to change the default target by creating a symbolic link to the target.

Procedure

1. To determine the default target unit:

```
# ls /usr/lib/systemd/system/default.target -l
```

2. To create a symbolic link:

```
# ln -sf /usr/lib/systemd/system/graphical.target /etc/systemd/system/default.target
```

3. Reboot the system:

```
# reboot
```

Verification steps

- Verify the newly created default.target:

```
$ systemctl get-default
multi-user.target
```

= Changing the current target

This procedure explains how to change the target unit in the current session using the `systemctl` command.

Procedure

- To change to a different target unit in the current session:

```
# systemctl isolate multi-user.target
```

This command starts the target unit named *multi-user* and all dependent units, and immediately stops all others.

Replace *multi-user* with the name of the target unit you want to use by default.

Verification steps

- Verify the newly created default.target:

```
$ systemctl get-default
multi-user.target
```

15.2.2.1. Booting 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.

Procedure

- To change the current target and enter rescue mode in the current session:

```
# systemctl rescue
```

Broadcast message from root@localhost on pts/0 (Fri 2013-10-25 18:23:15 CEST):

The system is going down to rescue mode NOW!



NOTE

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 a message, run the following command with the **--no-wall** command-line option: **# systemctl --no-wall rescue**

15.2.2.2. Booting 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.

Procedure

- To change the current target and enter emergency mode:

```
# systemctl emergency
```



NOTE

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 the following command with the **--no-wall** command-line option: **# systemctl --no-wall emergency**

CHAPTER 16. SHUTTING DOWN, SUSPENDING, AND HIBERNATING THE SYSTEM

This section contains instructions about shutting down, suspending, or hibernating your operating system.

16.1. SYSTEM SHUTDOWN

To shut down the system, you can either use the **systemctl** utility directly, or call this utility through the **shutdown** command.

The advantage of using the **shutdown** command is:

- The support for time argument
This is particularly useful for scheduled maintenance. Also, users have more time to react to the warning that a system shutdown has been scheduled.
- The option to cancel the shutdown

Additional resources

- [Shutting down the system using the shutdown command](#)
- [Shutting down the system using the systemctl command](#)
- [Overview of the power management commands with systemctl](#)

16.2. SHUTTING DOWN THE SYSTEM USING THE SHUTDOWN COMMAND

By following this procedure, you can use the **shutdown** command to perform various operations. You can either shut down the system and power off the machine at a certain time, or shut down and halt the system without powering off the machine, or cancel a pending shutdown.

Prerequisites

- Switch to the **root** user

Procedure

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

```
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 *wall* message can be appended to the command.

Alternatively, to shut down and halt the system after a delay, without powering off the machine, use:

```
shutdown --halt +m
```

Where *+m* is the delay time in minutes. The **now** keyword is an alias for **+0**.

To cancel a pending shutdown, use:

```
shutdown -c
```

Additional resources

- **shutdown(8)** manual page
- [Shutting down the system using the systemctl command](#)

16.3. SHUTTING DOWN THE SYSTEM USING THE SYSTEMCTL COMMAND

By following this procedure, you can use the **systemctl** command to perform various operations. You can either shut down the system and power off the machine, or shut down and halt the system without powering off the machine.

Prerequisites

- Switch to the **root** user

Procedure

- To shut down the system and power off the machine, use the command in the following format:

```
systemctl poweroff
```

Alternatively, to shut down and halt the system without powering off the machine, use:

```
systemctl halt
```



NOTE

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.

Additional resources

- [Shutting down the system using the shutdown command](#)

16.4. RESTARTING THE SYSTEM

You can restart the system by following this procedure.

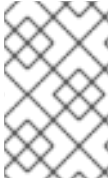
Prerequisites

- Switch to the **root** user

Procedure

- To restart the system, run the following command:

```
systemctl reboot
```



NOTE

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.

16.5. SUSPENDING THE SYSTEM

You can suspend the system by following this procedure.

Prerequisites

- Switch to the **root** user.

Procedure

- To suspend the system, run the following command:

```
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 from hibernation. However, note that the suspended system state is also vulnerable to power outages.

Additional resources

- [Hibernating the system](#)

16.6. HIBERNATING THE SYSTEM

By following this procedure, you can either hibernate the system, or hibernate and suspend the system.

Prerequisites

- Switch to the **root** user.

Procedure

- To hibernate the system, run the following command:

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. However, as a consequence, restoring the system from hibernation is significantly slower than restoring it from suspend mode.

Alternatively, to hibernate and suspend the system, run the following command:

systemctl hybrid-sleep

Additional resources

- [Suspending the system](#)

16.7. OVERVIEW OF THE POWER MANAGEMENT COMMANDS WITH SYSTEMCTL

You can use the following list of the **systemctl** commands to control the power management of your system.

Table 16.1. Overview of the systemctl power management commands

systemctl command	Description
systemctl halt	Halts the system.
systemctl poweroff	Powers off the system.
systemctl reboot	Restarts the system.
systemctl suspend	Suspends the system.
systemctl hibernate	Hibernates the system.
systemctl hybrid-sleep	Hibernates and suspends the system.

CHAPTER 17. WORKING WITH SYSTEMD UNIT FILES

This chapter includes the description of **systemd** unit files. The following sections show you how to:

- Create custom unit files
- Convert SysV init scripts to unit files
- Modify existing unit files
- Work with instantiated units

17.1. INTRODUCTION TO 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. [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. For a complete list of unit types, see [systemd unit files](#)

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 [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 [Working with instantiated units](#).

17.2. 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 [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 [Important \[Install\] section options](#).

Additional resources

- [Important\[Unit\] section options](#)
- [Important\[Service\] section options](#)
- [Important\[Install\] section options](#)

17.3. IMPORTANT [UNIT] SECTION OPTIONS

The following tables lists important options of the [Unit] section.

Table 17.1. Important [Unit] section options

Option ^[a]	Description
Description	A meaningful description of the unit. This text is displayed for example in the output of the systemctl status command.
Documentation	Provides a list of URIs referencing documentation for the unit.
After ^[b]	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 .
Requires	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.
Wants	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.
Conflicts	Configures negative dependencies, an opposite to Requires .
<p>[a] For a complete list of options configurable in the [Unit] section, see the systemd.unit(5) manual page.</p> <p>[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.</p>	

17.4. IMPORTANT SERVICE SECTION OPTIONS

The following tables lists important options of the [Service] section.

Table 17.2. Important [Service] section options

Option ^[a]	Description
Type	<p>Configures the unit process startup type that affects the functionality of ExecStart and related options. One of:</p> <ul style="list-style-type: none"> * simple – The default value. The process started with ExecStart is the main process of the service. * forking – The process started with ExecStart spawns a child process that becomes the main process of the service. The parent process exits when the startup is complete. * oneshot – This type is similar to simple, but the process exits before starting consequent units. * dbus – This type is similar to simple, but consequent units are started only after the main process gains a D-Bus name. * notify – This type is similar to simple, but consequent units are started only after a notification message is sent via the <code>sd_notify()</code> function. * idle – similar to simple, the actual execution of the service binary is delayed until all jobs are finished, which avoids mixing the status output with shell output of services.
ExecStart	<p>Specifies commands or scripts to be executed when the unit is started. ExecStartPre and ExecStartPost specify custom commands to be executed before and after ExecStart. Type=oneshot enables specifying multiple custom commands that are then executed sequentially.</p>
ExecStop	<p>Specifies commands or scripts to be executed when the unit is stopped.</p>
ExecReload	<p>Specifies commands or scripts to be executed when the unit is reloaded.</p>
Restart	<p>With this option enabled, the service is restarted after its process exits, with the exception of a clean stop by the systemctl command.</p>

Option [a]	Description
RemainAfterExit	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 Type=oneshot is configured.
[a] For a complete list of options configurable in the [Service] section, see the systemd.service(5) manual page.	

17.5. IMPORTANT [INSTALL] SECTION OPTIONS

The following tables lists important options of the [Install] section.

Table 17.3. Important [Install] section options

Option [a]	Description
Alias	Provides a space-separated list of additional names for the unit. Most systemctl commands, excluding systemctl enable , can use aliases instead of the actual unit name.
RequiredBy	A list of units that depend on the unit. When this unit is enabled, the units listed in RequiredBy gain a Require dependency on the unit.
WantedBy	A list of units that weakly depend on the unit. When this unit is enabled, the units listed in WantedBy gain a Want dependency on the unit.
Also	Specifies a list of units to be installed or uninstalled along with the unit.
DefaultInstance	Limited to instantiated units, this option specifies the default instance for which the unit is enabled. See Working with instantiated units
[a] For a complete list of options configurable in the [Install] section, see the systemd.unit(5) manual page.	

17.6. 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](#)

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](#).

Procedure

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**:

```
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 [Unit file structure](#).

The following is an example unit configuration for a network-related service:

```
[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 [Important \[Service\] section options](#).
 - **WantedBy** states the target or targets that the service should be started under. Think of these targets as of a replacement of the older concept of runlevels.
4. Notify **systemd** that a new ***name.service*** file exists by executing the following command as **root**:

—

```
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.

17.7. CREATING A CUSTOM UNIT FILE BY USING THE 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.

Procedure

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:

```
# cp /usr/lib/systemd/system/sshd.service /etc/systemd/system/sshd-second.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/ssh
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
```

7. Verify if the `sshd-second.service` is running by using the **systemctl status** command.
8. 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`.

17.8. 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.

17.9. FINDING THE SYSTEMD SERVICE DESCRIPTION

You can 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.

17.10. FINDING THE SYSTEMD SERVICE DEPENDENCIES

The LSB header might contain several directives that form dependencies between services. Most of them are translatable to systemd unit options, see the following table:

Table 17.4. Dependency options from the LSB header

LSB Option	Description	Unit File Equivalent
Provides	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.	–

LSB Option	Description	Unit File Equivalent
Required-Start	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 postfix , the Required-Start dependency on <code>\$network</code> was translated to the After dependency on <code>network.target</code> .	After, Before
Should-Start	Constitutes weaker dependencies than Required-Start. Failed Should-Start dependencies do not affect the service startup.	After, Before
Required-Stop, Should-Stop	Constitute negative dependencies.	Conflicts

17.11. 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 `multiuser.target` and `graphical.target`. Note that the `graphical.target` depends on `multiuser.target`, therefore it is not necessary to specify both. 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.

17.12. 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.

```
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 2>/dev/null 1>&2 && 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 **ExecStart**, **ExecStartPre**, **ExecStartPost**, **ExecStop**, and **ExecReload** options. The **/usr/sbin/postfix** together with supporting scripts are executed on service start. 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.

17.13. 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.

Procedure

- 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](#)
 - 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.
- To return to the default configuration of the unit, 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

- 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](#) [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**

17.14. EXTENDING THE DEFAULT UNIT CONFIGURATION

This section describes how to extend the default unit file with additional configuration options.

Procedure

1. 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.

2. 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 [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.

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

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

Example 17.1. 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.

1. Create a directory and a custom configuration file:

```
# mkdir /etc/systemd/system/httpd.service.d/
```

```
# touch /etc/systemd/system/httpd.service.d/custom_script.conf
```

2. 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:

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

3. To apply the unit changes, execute:

```
# 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.

17.15. OVERRIDING THE DEFAULT UNIT CONFIGURATION

This section describes how to override the default unit configuration.

Procedure

1. 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**:

```
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.

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

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

17.16. 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:

Procedure

1. Copy the **httpd** unit file to the **/etc/systemd/system/** directory:

```
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:

```
...
[Service]
...
PrivateTmp=true
TimeoutStartSec=10

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

3. Reload the **systemd** daemon:

```
systemctl daemon-reload
```

4. **Optional.** Verify the new timeout value:

```
systemctl show httpd -p TimeoutStartUsec
```



NOTE

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

17.17. MONITORING OVERRIDDEN UNITS

This section describes how to display an overview of overridden or modified unit files.

Procedure

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

```
systemd-delta
```

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

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

--- /usr/lib/systemd/system/autofs.service    2014-10-16 21:30:39.000000000 -0400
+ /etc/systemd/system/autofs.service  2014-11-21 10:00:58.513568275 -0500
@@ -8,7 +8,8 @@
EnvironmentFile=-/etc/sysconfig/autofs
ExecStart=/usr/sbin/automount $OPTIONS --pid-file /run/autofs.pid
ExecReload=/usr/bin/kill -HUP $MAINPID
-TimeoutSec=180
```

```

+TimeoutSec=240
+Restart=Always

[Install]
WantedBy=multi-user.target

[MASKED]    /etc/systemd/system/cups.service → /usr/lib/systemd/system/cups.service
[EXTENDED]  /usr/lib/systemd/system/sss.service →
/etc/systemd/system/sss.service.d/journal.conf

4 overridden configuration files found.

```

17.18. 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.

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

```

[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**.

17.19. IMPORTANT UNIT SPECIFIERS

Wildcard characters, called **unit specifiers**, can be used in any unit configuration file. Unit specifiers substitute certain unit parameters and are interpreted at runtime. The following table lists unit specifiers that are particularly useful for template units.

Table 17.5. Important unit specifiers

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

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

17.20. ADDITIONAL RESOURCES

- [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](#)
- [Is there any useful information about writing unit files?](#)
- [How to set limits for services in RHEL 7 and systemd](#)

CHAPTER 18. 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.

18.1. 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:

Procedure

```
$ 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 18.1. The output of the `systemd-analyze critical-chain` command

```
[admin@localhost ~]$ systemd-analyze critical-chain
The time after the unit is active or started is printed after the "@" character.
The time the unit takes to start is printed after the "+" character.

graphical.target @19.706s
├─multi-user.target @19.706s
│   └─tuned.service @5.616s +3.397s
│       └─network.target @5.614s
│           └─wpa_supplicant.service @16.025s +125ms
│               └─dbus.service @2.461s
│                   └─basic.target @2.444s
│                       └─sockets.target @2.444s
│                           └─iscsiuio.socket @2.444s
│                               └─sysinit.target @2.431s
│                                   └─systemd-update-utmp.service @2.419s +10ms
│                                       └─auditd.service @2.292s +126ms
│                                           └─systemd-tmpfiles-setup.service @2.228s +63ms
│                                               └─import-state.service @2.171s +54ms
│                                                   └─local-fs.target @2.168s
│                                                       └─run-user-42.mount @9.536s
│                                                           └─local-fs-pre.target @2.112s
│                                                               └─lvm2-monitor.service @2.087s +25ms
│                                                                   └─dm-event.socket @968ms
│                                                                       └─.mount
│                                                                           └─system.slice
│                                                                               └─.slice

[admin@localhost ~]$
```

18.2. 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:

```
# systemctl disable service_name
```

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, 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 18.1. Services enabled by default on a minimal installation of RHEL

Service name	Can it be disabled?	More information
auditd.service	yes	Disable auditd.service only if you do not need audit messages from the kernel. Be aware that if you disable auditd.service , the /var/log/audit/audit.log 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 systemctl disable auditd command, you only disable the service, but not the kernel part. To disable system auditing in its entirety, set audit=0 on kernel command line.
autovt@.service	no	This service runs only when it is really needed, so it does not need to be disabled.
crond.service	yes	Be aware that no items from crontab will run if you disable crond.service.
dbus-org.fedoraproject.FirewallD1.service	yes	A symlink to firewalld.service
dbus-org.freedesktop.NetworkManager.service	yes	A symlink to NetworkManager.service
dbus-org.freedesktop.nm-dispatcher.service	yes	A symlink to NetworkManager-dispatcher.service
firewalld.service	yes	Disable firewalld.service only if you do not need firewall.
getty@.service	no	This service runs only when it is really needed, so it does not need to be disabled.
import-state.service	yes	Disable import-state.service only if you do not need to boot from a network storage.
irqbalance.service	yes	Disable irqbalance.service only if you have just one CPU. Do not disable irqbalance.service on systems with multiple CPUs.

Service name	Can it be disabled?	More information
kdump.service	yes	Disable kdump.service only if you do not need reports from kernel crashes.
loadmodules.service	yes	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 installation.
lvm2-monitor.service	yes	Disable lvm2-monitor.service only if you do not use Logical Volume Manager (LVM).
microcode.service	no	Do not be disable the service because it provides updates of the microcode software in CPU.
NetworkManager-dispatcher.service	yes	Disable NetworkManager-dispatcher.service only if you do not need notifications on network configuration changes (for example in static networks).
NetworkManager-wait-online.service	yes	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.
NetworkManager.service	yes	Disable NetworkManager.service only if you do not need connection to a network.
nis-domainname.service	yes	Disable nis-domainname.service only if you do not use Network Information Service (NIS).
rhsmcertd.service	no	
rngd.service	yes	Disable rngd.service 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).
rsyslog.service	yes	Disable rsyslog.service only if you do not need persistent logs, or you set systemd-journal to persistent mode.

Service name	Can it be disabled?	More information
selinux-autorelabel-mark.service	yes	Disable selinux-autorelabel-mark.service only if you do not use SELinux.
sshd.service	yes	Disable sshd.service only if you do not need remote logins by OpenSSH server.
sssd.service	yes	Disable sssd.service 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 sssd-* units if you disable sssd.service .
syslog.service	yes	An alias for rsyslog.service
tuned.service	yes	Disable tuned.service only if you do need to use performance tuning.
lvm2-lvmpolld.socket	yes	Disable lvm2-lvmpolld.socket only if you do not use Logical Volume Manager (LVM).
dnf-makecache.timer	yes	Disable dnf-makecache.timer only if you do not need your package metadata to be updated automatically.
unbound-anchor.timer	yes	Disable unbound-anchor.timer 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.

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

```
$ systemctl cat <service_name>
```

```
$ systemctl help <service_name>
```

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.

18.3. ADDITIONAL RESOURCES

- **systemctl**(1) man page
- **systemd**(1) man page
- **systemd-delta**(1) man page
- **systemd.directives**(7) man page
- **systemd.unit**(5) man page
- **systemd.service**(5) man page
- **systemd.target**(5) man page
- **systemd.kill**(5) man page
- [systemd Home Page](#)

CHAPTER 19. INTRODUCTION TO MANAGING USER AND GROUP ACCOUNTS

The control of users and groups is a core element of Red Hat Enterprise Linux (RHEL) system administration. Each RHEL user has distinct login credentials and can be assigned to various groups to customize their system privileges.

19.1. INTRODUCTION TO USERS AND GROUPS

A user who creates a file is the owner of that file *and* the group owner of that file. The file is assigned separate read, write, and execute permissions for the owner, the group, and those outside that group. The file owner can be changed only by the **root** user. Access permissions to the file can be changed by both the **root** user and the file owner. A regular user can change group ownership of a file they own to a group of which they are a member of.

Each user is associated with a unique numerical identification number called *user ID (UID)*. Each group is associated with a *group ID (GID)*. Users within a group share the same permissions to read, write, and execute files owned by that group.

19.2. CONFIGURING RESERVED USER AND GROUP IDS

RHEL reserves user and group IDs below 1000 for system users and groups. You can find the reserved user and group IDs in the **setup** package. To view reserved user and group IDs, use:

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

It is recommended to assign IDs to the new users and groups starting at 5000, as the reserved range can increase in the future.

To make the IDs assigned to new users start at 5000 by default, modify the **UID_MIN** and **GID_MIN** parameters in the **/etc/login.defs** file.

Procedure

To modify make the IDs assigned to new users start at 5000 by default, use:

1. Open the **/etc/login.defs** file in an editor of your choice.
2. Find the lines that define the minimum value for automatic UID selection.

```
# Min/max values for automatic uid selection in useradd
#
UID_MIN          1000
```

3. Modify the **UID_MIN** value to start at 5000.

```
# Min/max values for automatic uid selection in useradd
#
UID_MIN          5000
```

4. Find the lines that define the minimum value for automatic GID selection.

```
# Min/max values for automatic gid selection in groupadd
#
GID_MIN          1000
```

Note that for users and groups created before you changed the **UID_MIN** and **GID_MIN** values, UIDs and GIDs still start at the default 1000.



WARNING

Do not raise IDs reserved by the system above 1000 by changing **SYS_UID_MAX** to avoid conflict with systems that retain the 1000 limit.

19.3. USER PRIVATE GROUPS

RHEL uses the *user private group* (**UPG**) system configuration, which makes UNIX groups easier to manage. A user private group is created whenever a new user is added to the system. The user private group has the same name as the user for which it was created and that user is the only member of the user private group.

UPGs simplify the collaboration on a project between multiple users. In addition, UPG system configuration makes it safe to set default permissions for a newly created file or directory, as it allows both the user, and the group this user is a part of, to make modifications to the file or directory.

A list of all groups is stored in the **/etc/group** configuration file.

CHAPTER 20. MANAGING USER ACCOUNTS IN THE WEB CONSOLE

The RHEL web console offers a graphical interface that enables you to execute a wide range of administrative tasks without accessing your terminal directly. For example, you can add, edit or remove system user accounts.

After reading this section, you will know:

- From where the existing accounts come from.
- How to add new accounts.
- How to set password expiration.
- How and when to terminate user sessions.

Prerequisites

- Set up the RHEL web console. For details, see [Getting started using the RHEL web console](#) ,
- Log in to the RHEL web console with an account that has administrator permissions assigned. For details, see [Logging in to the RHEL web console](#) .

20.1. SYSTEM USER ACCOUNTS MANAGED IN THE WEB CONSOLE

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

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

The RHEL 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.

After logging into the RHEL web console, you can perform the following operations:

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

20.2. ADDING NEW ACCOUNTS USING THE WEB CONSOLE

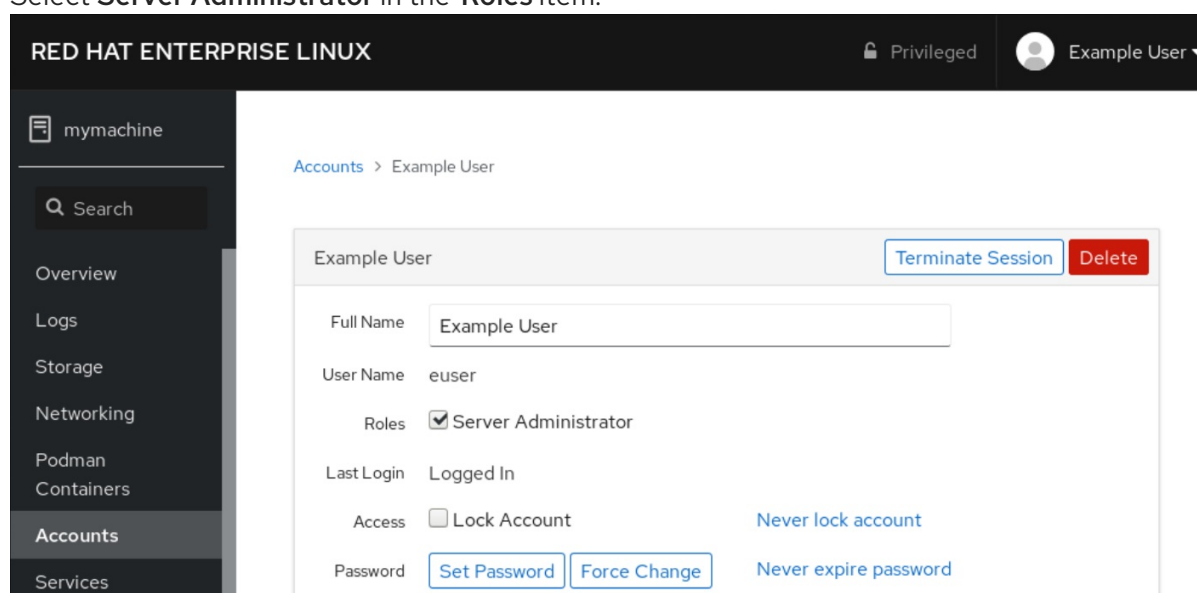
Use the following steps for adding user accounts to the system and setting administration rights to the accounts through the RHEL web console.

Prerequisites

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

Procedure

1. Log in to the RHEL web console.
2. Click **Accounts**.
3. Click **Create New Account**.
 1. 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.
 2. 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.
 1. Click **Create** to save the settings and close the dialog box.
 2. Select the newly created account.
 3. 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.

20.3. ENFORCING PASSWORD EXPIRATION IN THE WEB CONSOLE

By default, user accounts have set passwords to never expire. You can set system passwords to expire after a defined number of days. When the password expires, the next login attempt will prompt for a password change.

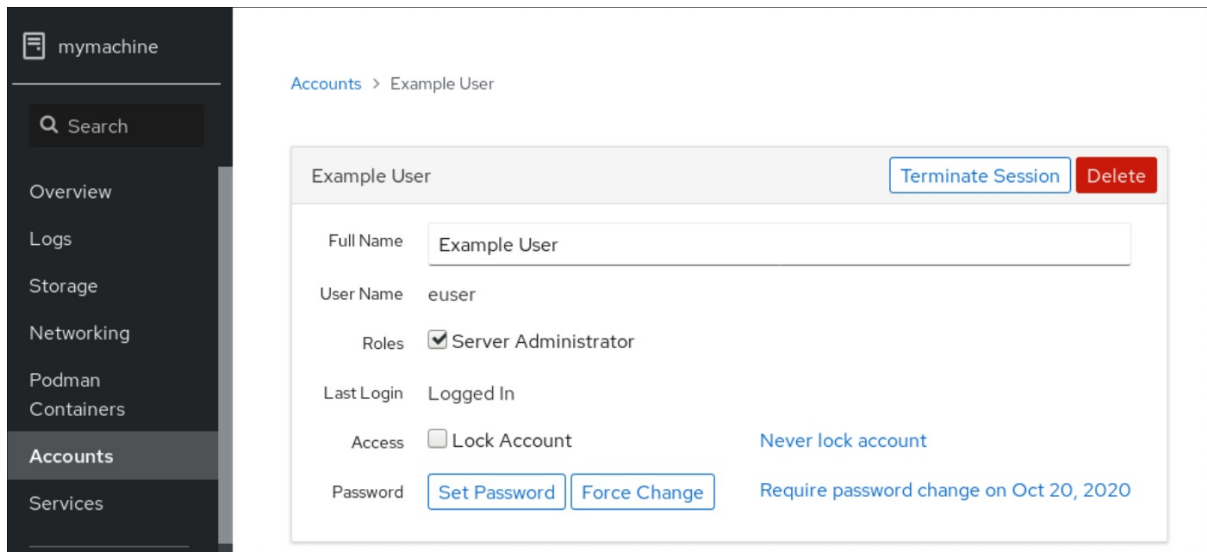
Procedure

1. Log in to the RHEL 8 web console.
2. Click **Accounts**.

3. Select the user account for which to enforce password expiration.
4. In the user account settings, click **Never expire password**.
5. In the **Password Expiration** dialog box, select **Require password change every ... days** and enter a positive whole number representing the number of days when the password expires.
1. Click **Change**.

Verification steps

- To verify that the password expiration is set, open the account settings. The RHEL 8 web console displays a link with the date of expiration.



20.4. TERMINATING USER SESSIONS IN THE WEB CONSOLE

A user creates user sessions when logging into the system. Terminating user sessions means to log the user out from the system. It can be helpful if you need to perform administrative tasks sensitive to configuration changes, for example, system upgrades.

In each user account in the RHEL 8 web console, you can terminate all sessions for the account except for the web console session you are currently using. This prevents you from losing access to your system.

Procedure

1. Log in to the RHEL 8 web console.
2. Click **Accounts**.
3. Click the user account for which you want to terminate the session.
4. Click **Terminate Session**.
If the **Terminate Session** button is inactive, the user is not logged in to the system.

The RHEL web console terminates the sessions.

CHAPTER 21. MANAGING USERS FROM THE COMMAND LINE

You can manage users and groups using the command-line interface (CLI). This enables you to add, remove, and modify users and user groups in Red Hat Enterprise Linux environment.

21.1. ADDING A NEW USER FROM THE COMMAND LINE

This section describes how to use the **useradd** utility to add a new user.

Prerequisites

- **Root** access

Procedure

- To add a new user, use:

```
# useradd options username
```

Replace *options* with the command-line options for the **useradd** command, and replace *username* with the name of the user.

Example 21.1. Adding a new user

To add the user **sarah** with user ID **5000**, use:

+

```
# useradd -u 5000 sarah
```

Verification steps

- To verify the new user is added, use the **id** utility.

```
# id sarah
```

The output returns:

```
uid=5000(sarah) gid=5000(sarah) groups=5000(sarah)
```

Additional resources

- **useradd** man page

21.2. ADDING A NEW GROUP FROM THE COMMAND LINE

This section describes how to use the **groupadd** utility to add a new group.

Prerequisites

- **Root** access

Procedure

- To add a new group, use:

```
# groupadd options group-name
```

Replace *options* with the command-line options for the **groupadd** command, and replace *group-name* with the name of the group.

Example 21.2. Adding a new group

To add the group **sysadmins** with group ID **5000**, use:

+

```
# groupadd -g 5000 sysadmins
```

Verification steps

- To verify the new group is added, use the **tail** utility.

```
# tail /etc/group
```

The output returns:

```
sysadmins:x:5000:
```

Additional resources

- **groupadd** man page

21.3. ADDING A USER TO A SUPPLEMENTARY GROUP FROM THE COMMAND LINE

You can add a user to a supplementary group to manage permissions or enable access to certain files or devices.

Prerequisites

- **root** access

Procedure

- To add a group to the supplementary groups of the user, use:

```
# usermod --append -G group-name username
```

Replace *group-name* with the name of the group, and replace *username* with the name of the user.

Example 21.3. Adding a user to a supplementary group

To add the user **sysadmin** to the group **system-administrators**, use:

```
# usermod --append -G system-administrators sysadmin
```

Verification steps

- To verify the new groups is added to the supplementary groups of the user **sysadmin**, use:

```
# groups sysadmin
```

The output displays:

```
sysadmin : sysadmin system-administrators
```

21.4. CREATING A GROUP DIRECTORY

Under the UPG system configuration, you can apply the *set-group identification permission* (**setgid** bit) to a directory. The **setgid** bit makes managing group projects that share a directory simpler. When you apply the **setgid** bit to a directory, files created within that directory are automatically assigned to a group that owns the directory. Any user that has the permission to write and execute within this group can now create, modify, and delete files in the directory.

The following section describes how to create group directories.

Prerequisites

- **Root** access

Procedure

1. Create a directory:

```
# mkdir directory-name
```

Replace *directory-name* with the name of the directory.

2. Create a group:

```
# groupadd group-name
```

Replace *group-name* with the name of the group.

3. Add users to the group:

```
# usermod --append -G group-name username
```

Replace *group-name* with the name of the group, and replace *username* with the name of the user.

4. Associate the user and group ownership of the directory with the *group-name* group:

```
# chown :group-name directory-name
```

Replace *group-name* with the name of the group, and replace *directory-name* with the name of the directory.

5. Set the write permissions to allow the users to create and modify files and directories and set the **setgid** bit to make this permission be applied within the *directory-name* directory:

```
# chmod g+rxws directory-name
```

Replace *directory-name* with the name of the directory.

Now all members of the ***group-name*** group can create and edit files in the ***directory-name*** directory. Newly created files retain the group ownership of ***group-name*** group.

Verification steps

- To verify the correctness of set permissions, use:

```
# ls -ld directory-name
```

Replace *directory-name* with the name of the directory.

The output returns:

```
drwxrwsr-x. 2 root group-name 6 Nov 25 08:45 directory-name
```

CHAPTER 22. EDITING USER GROUPS USING THE COMMAND LINE

A user belongs to a certain set of groups that allow a logical collection of users with a similar access to files and folders. You can edit the primary and supplementary user groups from the command line to change the user's permissions.

22.1. PRIMARY AND SUPPLEMENTARY USER GROUPS

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

On Linux, user groups can act as primary or supplementary. Primary and supplementary groups have the following properties:

Primary group

- Every user has just one primary group at all times.
- You can change the user's primary group.

Supplementary groups

- You can add an existing user to an existing supplementary group to manage users with the same security and access privileges within the group.
- Users can be members of zero or multiple supplementary groups.

22.2. LISTING THE PRIMARY AND SUPPLEMENTARY GROUPS OF A USER

You can list the groups of users to see which primary and supplementary groups they belong to.

Procedure

- Display the names of the primary and any supplementary group of a user:

```
$ groups user-name
```

Replace *user-name* with the name of the user. If you do not provide a user name, the command displays the group membership for the current user. The first group is the primary group followed by the optional supplementary groups.

Example 22.1. Listing of groups for user sarah:

```
$ groups sarah
```

The output displays:

```
sarah : sarah wheel developer
```

User **sarah** has a primary group **sarah** and is a member of supplementary groups **wheel** and **developer**.

Example 22.2. Listing of groups for user marc:

```
$ groups marc
```

The output displays:

```
marc : marc
```

User **marc** has only a primary group **marc** and no supplementary groups.

22.3. CHANGING THE PRIMARY GROUP OF A USER

You can change the primary group of an existing user to a new group.

Prerequisites:

1. **root** access
2. The new group must exist

Procedure

- Change the primary group of a user:

```
# usermod -g group-name user-name
```

Replace *group-name* with the name of the new primary group, and replace *user-name* with the name of the user.



NOTE

When you change a user's primary group, the command also automatically changes the group ownership of all files in the user's home directory to the new primary group. You must fix the group ownership of files outside of the user's home directory manually.

Example 22.3. Example of changing the primary group of a user:

If the user **sarah** belongs to the primary group **sarah1**, and you want to change the primary group of the user to **sarah2**, use:

```
# usermod -g sarah2 sarah
```

Verification steps

- Verify that you changed the primary group of the user:

```
$ groups sarah
```

The output displays:

```
sarah : sarah2
```

22.4. ADDING A USER TO A SUPPLEMENTARY GROUP FROM THE COMMAND LINE

You can add a user to a supplementary group to manage permissions or enable access to certain files or devices.

Prerequisites

- **root** access

Procedure

- To add a group to the supplementary groups of the user, use:

```
# usermod --append -G group-name username
```

Replace *group-name* with the name of the group, and replace *username* with the name of the user.

Example 22.4. Adding a user to a supplementary group

To add the user **sysadmin** to the group **system-administrators**, use:

```
# usermod --append -G system-administrators sysadmin
```

Verification steps

- To verify the new groups is added to the supplementary groups of the user **sysadmin**, use:

```
# groups sysadmin
```

The output displays:

```
sysadmin : sysadmin system-administrators
```

22.5. REMOVING A USER FROM A SUPPLEMENTARY GROUP

You can remove an existing user from a supplementary group to limit their permissions or access to files and devices.

Prerequisites

- **root** access

Procedure

- Remove a user from a supplementary group:

```
# gpasswd -d user-name group-name
```

Replace *user-name* with the name of the user, and replace *group-name* with the name of the supplementary group.

Example 22.5. Removing user from a supplementary group

If the user sarah has a primary group **sarah2**, and belongs to the secondary groups **wheel** and **developers**, and you want to remove that user from the group **developers**, use:

```
# gpasswd -d sarah developers
```

Verification steps

- Verify that you removed the user sarah from the secondary group developers:

```
$ groups sarah
```

The output displays:

```
sarah : sarah2 wheel
```

22.6. CHANGING ALL OF THE SUPPLEMENTARY GROUPS OF A USER

You can overwrite the list of supplementary groups that you want the user to remain a member of.

Prerequisites

- **root** access
- The supplementary groups must exist

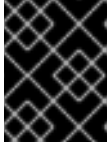
Procedure

- Overwrite a list of user's supplementary groups:

```
# usermod -G group-names username
```

Replace *group-names* with the name of one or more supplementary groups. To add the user to several supplementary groups at once, separate the group names using commas and no intervening spaces. For example: **wheel,developer**.

Replace *user-name* with the name of the user.



IMPORTANT

If the user is currently a member of a group that you do not specify, the command removes the user from the group.

Example 22.6. Changing the list of supplementary groups of a user

If the user **sarah** has a primary group **sarah2**, and belongs to the supplementary group **wheel**, and you want the user to belong to three more supplementary groups **developer**, **sysadmin**, and **security**, use:

```
# usermod -G wheel,developer,sysadmin,security sarah
```

Verification steps

- Verify that you set the list of the supplementary groups correct:

```
# groups sarah
```

The output displays:

```
sarah : sarah2 wheel developer sysadmin security
```

CHAPTER 23. MANAGING SUDO ACCESS

System administrators can grant **sudo** access to allow non-root users to execute administrative commands that are normally reserved for the **root** user. As a result, non-root users can execute such commands without logging in to the **root** user account.

23.1. USER AUTHORIZATIONS IN SUDOERS

The **/etc/sudoers** file specifies which users can run which commands using the **sudo** command. The rules can apply to individual users and user groups. You can also use aliases to simplify defining rules for groups of hosts, commands, and even users. Default aliases are defined in the first part of the **/etc/sudoers** file.

When a user tries to use **sudo** privileges to run a command that is not allowed in the **/etc/sudoers** file, the system records a message containing **username : user NOT in sudoers** to the journal log.

The default **/etc/sudoers** file provides information and examples of authorizations. You can activate a specific example rule by removing the **#** comment character from the beginning of the line. The authorizations section relevant for user is marked with the following introduction:

```
## Next comes the main part: which users can run what software on
## which machines (the sudoers file can be shared between multiple
## systems).
```

You can use the following format to create new **sudoers** authorizations and to modify existing authorizations:

```
username hostname=path/to/command
```

Where:

- *username* is the name of the user or group, for example, **user1** or **%group1**.
- *hostname* is the name of the host on which the rule applies.
- *path/to/command* is the complete absolute path to the command. You can also limit the user to only performing a command with specific options and arguments by adding those options after the command path. If you do not specify any options, the user can use the command with all options.

You can replace any of these variables with **ALL** to apply the rule to all users, hosts, or commands.



WARNING

With overly permissive rules, such as **ALL ALL=(ALL) ALL**, all users are able to run all commands as all users on all hosts. This can lead to security risks.

You can specify the arguments negatively using the **!** operator. For example, use **!root** to specify all users except the **root** user. Note that using the allowlists to allow specific users, groups, and commands,

is more secure than using the blocklists to disallowing specific users, groups, and commands. By using the allowlists you also block new unauthorized users or groups.



WARNING

Avoid using negative rules for commands because users can overcome such rules by renaming commands using the **alias** command.

The system reads the **/etc/sudoers** file from beginning to end. Therefore, if the file contains multiple entries for a user, the entries are applied in order. In case of conflicting values, the system uses the last match, even if it is not the most specific match.

The preferred way of adding new rules to **sudoers** is by creating new files in the **/etc/sudoers.d/** directory instead of entering rules directly to the **/etc/sudoers** file. This is because the contents of this directory are preserved during system updates. In addition, it is easier to fix any errors in the separate files than in the **/etc/sudoers** file. The system reads the files in the **/etc/sudoers.d** directory when it reaches the following line in the **/etc/sudoers** file:

```
#includedir /etc/sudoers.d
```

Note that the number sign **#** at the beginning of this line is part of the syntax and does not mean the line is a comment. The names of files in that directory must not contain a period **.** and must not end with a tilde **~**.

23.2. GRANTING SUDO ACCESS TO A USER

System administrators can grant **sudo** access to allow non-root users to execute administrative commands. The **sudo** command provides users with administrative access without using the password of the **root** user.

When users need to perform an administrative command, they can precede that command with **sudo**. The command is then executed as if they were the **root** user.

Be aware of the following limitations:

- Only users listed in the **/etc/sudoers** configuration file can use the **sudo** command.
- The command is executed in the shell of the user, not in the **root** shell.

Prerequisites

- **root** access

Procedure

1. As root, open the **/etc/sudoers** file.

```
# visudo
```

The **/etc/sudoers** file defines the policies applied by the **sudo** command.

2. In the **/etc/sudoers** file, find the lines that grant **sudo** access to users in the administrative **wheel** group.

```
## Allows people in group wheel to run all commands
%wheel    ALL=(ALL)    ALL
```

3. Make sure the line that starts with **%wheel** does not have the **#** comment character before it.
4. Save any changes, and exit the editor.
5. Add users you want to grant **sudo** access to into the administrative **wheel** group.

```
# usermod --append -G wheel username
```

Replace *username* with the name of the user.

Verification steps

- Verify that the user is added to the administrative **wheel** group:

```
# id username
uid=5000(username) gid=5000(_username) groups=5000(username),10(wheel)
```

23.3. ENABLING UNPRIVILEGED USERS TO RUN CERTAIN COMMANDS

You can configure a policy that allows unprivileged user to run certain command on a specific workstation. To configure this policy, you need to edit the **sudoers.d** file.

Prerequisites

- **root** access

Procedure

1. As root, create a new **sudoers.d** directory under **/etc/**:

```
# mkdir -p /etc/sudoers.d/
```

2. Create a new file in the **/etc/sudoers.d** directory:

```
# visudo -f /etc/sudoers.d/file-name
```

Replace *file-name* with the name of the file you want to create. The file will open automatically.

3. Add the following line to the newly created file:

```
username hostname = /path/to/the/command
```

Replace *username* with the name of the user. Replace *hostname* with the name of the host. Replace */path/to/the/command* with the absolute path to the command (for example, **/usr/bin/yum**).

4. Save any changes, and exit the editor.

Example 23.1. Enabling an unprivileged user to install programs with yum and dnf

To enable the user *sarah* to install programs on the **localhost.localdomain** workstation using the **yum** and **dnf** utilities with **sudo** privileges, use:

1. As root, create a new **sudoers.d** directory under **/etc/**:

```
# mkdir -p /etc/sudoers.d/
```

2. Create a new file in the **/etc/sudoers.d** directory:

```
# visudo -f /etc/sudoers.d/sarah
```

The file will open automatically.

3. Add the following line to the **/etc/sudoers.d/sarah** file:

```
sarah localhost.localdomain = /usr/bin/yum, /usr/bin/dnf
```

Ensure that the two command paths are separated by a **,** comma followed by a space.

4. *Optional:* To receive email notifications every time the user *sarah* attempts to use **sudo** privileges, add the following lines to the file:

```
Defaults    mail_always
Defaults    mailto="email@domain.com"
```

5. To verify if the user *sarah* can run the **yum** command with **sudo** privileges, switch the account:

```
# su sarah -
```

6. Enter the **sudo yum** command:

```
$ sudo yum
[sudo] password for sarah:
```

Enter the **sudo** password for the user *sarah*.

7. The system displays the list of **yum** commands and options:

```
...
usage: yum [options] COMMAND
...
```

If you receive the **sarah is not in the sudoers file. This incident will be reported.** message, the configuration was not completed correctly. Ensure that you are executing this procedure as **root** and that you followed the steps thoroughly.

23.4. ADDITIONAL RESOURCES

- The **sudo(8)** man page
- The **visudo(8)** man page

CHAPTER 24. CHANGING AND RESETTING THE ROOT PASSWORD

If the existing root password is no longer satisfactory or is forgotten, you can change or reset it both as the **root** user and a non-root user.

24.1. CHANGING THE ROOT PASSWORD AS THE ROOT USER

This section describes how to use the **passwd** command to change the **root** password as the **root** user.

Prerequisites

- **Root** access

Procedure

- To change the **root** password, use:

```
# passwd
```

You are prompted to enter your current password before you can change it.

24.2. CHANGING OR RESETTING THE FORGOTTEN ROOT PASSWORD AS A NON-ROOT USER

This section describes how to use the **passwd** command to change or reset the forgotten **root** password as a non-root user.

Prerequisites

- You are able to log in as a non-root user.
- You are a member of the administrative **wheel** group.

Procedure

- To change or reset the **root** password as a non-root user that belongs to the **wheel** group, use:

```
$ sudo passwd root
```

You are prompted to enter your current non-root password before you can change the **root** password.

24.3. RESETTING THE ROOT PASSWORD ON BOOT

If you are unable to log in as a non-root user or do not belong to the administrative **wheel** group, you can reset the root password on boot by switching into a specialized **chroot jail** environment.

Procedure

1. Reboot the system and, on the GRUB 2 boot screen, press the **e** key to interrupt the boot process.

The kernel boot parameters appear.

```
load_video
set gfx_payload=keep
insmod gzio
linux ($root)/vmlinuz-4.18.0-80.e18.x86_64 root=/dev/mapper/rhel-root ro crash\
kernel=auto resume=/dev/mapper/rhel-swap rd.lvm.lv/swap rhgb quiet
initrd ($root)/initramfs-4.18.0-80.e18.x86_64.img $tuned_initrd
```

2. Go to the end of the line that starts with **linux**.

```
linux ($root)/vmlinuz-4.18.0-80.e18.x86_64 root=/dev/mapper/rhel-root ro crash\
kernel=auto resume=/dev/mapper/rhel-swap rd.lvm.lv/swap rhgb quiet
```

Press **Ctrl+e** to jump to the end of the line.

3. Add **rd.break** to the end of the line that starts with **linux**.

```
linux ($root)/vmlinuz-4.18.0-80.e18.x86_64 root=/dev/mapper/rhel-root ro crash\
kernel=auto resume=/dev/mapper/rhel-swap rd.lvm.lv/swap rhgb quiet rd.break
```

4. Press **Ctrl+x** to start the system with the changed parameters.
The **switch_root** prompt appears.

5. Remount the file system as writable:

```
mount -o remount,rw /sysroot
```

The file system is mounted as read-only in the **/sysroot** directory. Remounting the file system as writable allows you to change the password.

6. Enter the **chroot** environment:

```
chroot /sysroot
```

The **sh-4.4#** prompt appears.

7. Reset the **root** password:

```
passwd
```

Follow the instructions displayed by the command line to finalize the change of the **root** password.

8. Enable the SELinux relabeling process on the next system boot:

```
touch /.autorelabel
```

9. Exit the **chroot** environment:

```
exit
```

10. Exit the **switch_root** prompt:

-

```
exit
```

11. Wait until the SELinux relabeling process is finished. Note that relabeling a large disk might take a long time. The system reboots automatically when the process is complete.

Verification steps

1. To verify that the **root** password is successfully changed, log in as a normal user and open the Terminal.
2. Run the interactive shell as root:

```
$ su
```

3. Enter your new **root** password.
4. Print the user name associated with the current effective user ID:

```
whoami
```

The output returns:

```
root
```

CHAPTER 25. MANAGING FILE PERMISSIONS

File permissions control the ability of user and group accounts to view, modify, access, and execute the contents of the files and directories.

Every file or directory has three levels of ownership:

- User owner (**u**).
- Group owner (**g**).
- Others (**o**).

Each level of ownership can be assigned the following permissions:

- Read (**r**).
- Write (**w**).
- Execute (**x**).

Note that the execute permission for a file allows you to execute that file. The execute permission for a directory allows you to access the contents of the directory, but not execute it.

When a new file or directory is created, the default set of permissions are automatically assigned to it. The default permissions for a file or directory are based on two factors:

- Base permission.
- The *user file-creation mode mask* (**umask**).

25.1. BASE FILE PERMISSIONS

Whenever a new file or directory is created, a base permission is automatically assigned to it. Base permissions for a file or directory can be expressed in *symbolic* or *octal* values.

Permission	Symbolic value	Octal value
No permission	---	0
Execute	--x	1
Write	-w-	2
Write and execute	-wx	3
Read	r--	4
Read and execute	r-x	5
Read and write	rw-	6

Read, write, execute	rwX	7
----------------------	-----	---

The base permission for a directory is **777 (drwxrwxrwx)**, which grants everyone the permissions to read, write, and execute. This means that the directory owner, the group, and others can list the contents of the directory, create, delete, and edit items within the directory, and descend into it.

Note that individual files within a directory can have their own permission that might prevent you from editing them, despite having unrestricted access to the directory.

The base permission for a file is **666 (-rw-rw-rw-)**, which grants everyone the permissions to read and write. This means that the file owner, the group, and others can read and edit the file.

Example 25.1. Permissions for a file

If a file has the following permissions:

```
$ ls -l
-rwxrw----. 1 sysadmins sysadmins 2 Mar 2 08:43 file
```

- **-** indicates it is a file.
- **rwX** indicates that the file owner has permissions to read, write, and execute the file.
- **rw-** indicates that the group has permissions to read and write, but not execute the file.
- **---** indicates that other users have no permission to read, write, or execute the file.
- **.** indicates that the SELinux security context is set for the file.

Example 25.2. Permissions for a directory

If a directory has the following permissions:

```
$ ls -dl directory
drwxr-----. 1 sysadmins sysadmins 2 Mar 2 08:43 directory
```

- **d** indicates it is a directory.
- **rwX** indicates that the directory owner has the permissions to read, write, and access the contents of the directory.
As a directory owner, you can list the items (files, subdirectories) within the directory, access the content of those items, and modify them.
- **r--** indicates that the group has permissions to read, but not write or access the contents of the directory.
As a member of the group that owns the directory, you can list the items within the directory. You cannot access information about the items within the directory or modify them.
- **---** indicates that other users have no permission to read, write, or access the contents of the directory.
As someone who is not an user owner, or as group owner of the directory, you cannot list the items within the directory, access information about those items, or modify them.

- . indicates that the SELinux security context is set for the directory.



NOTE

The base permission that is automatically assigned to a file or directory is **not** the default permission the file or directory ends up with. When you create a file or directory, the base permission is altered by the *umask*. The combination of the base permission and the *umask* creates the default permission for files and directories.

25.2. USER FILE-CREATION MODE MASK

The user file-creation mode mask (*umask*) is variable that controls how file permissions are set for newly created files and directories. The *umask* automatically removes permissions from the base permission value to increase the overall security of a linux system. The *umask* can be expressed in *symbolic* or *octal* values.

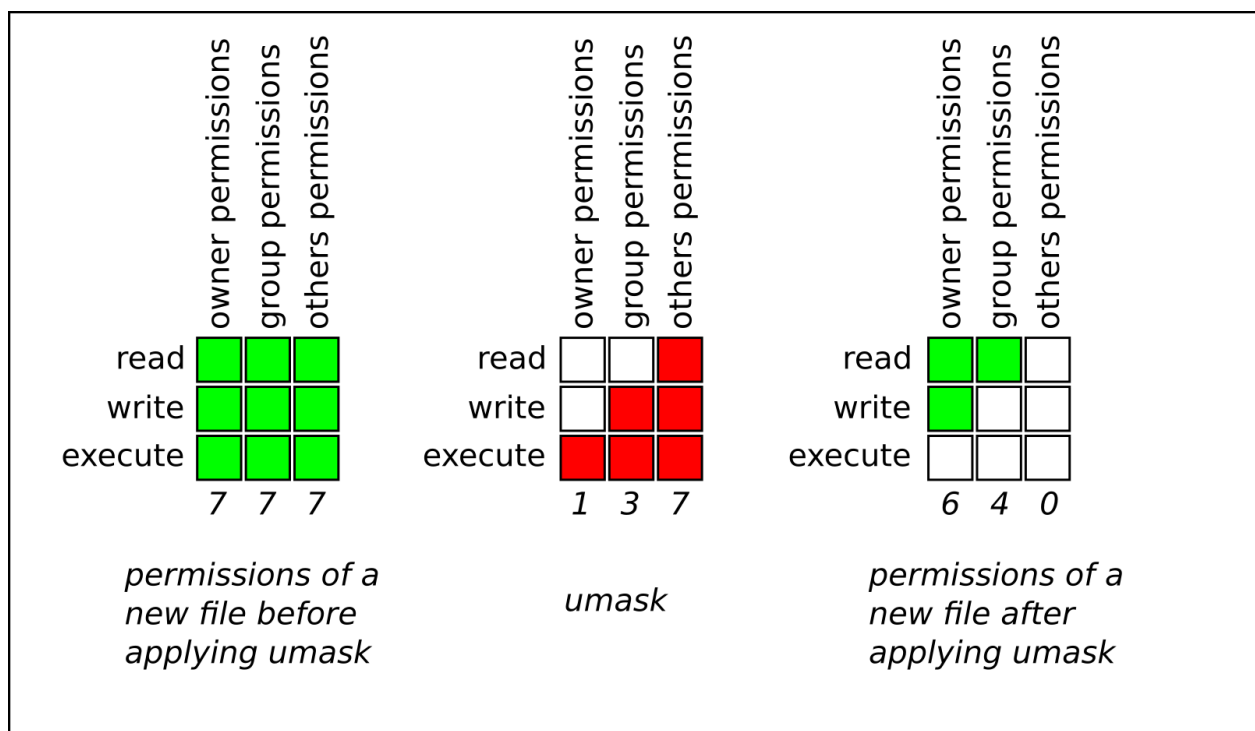
Permission	Symbolic value	Octal value
Read, write, and execute	rwX	0
Read and write	rw-	1
Read and execute	r-X	2
Read	r--	3
Write and execute	-wX	4
Write	-w-	5
Execute	--X	6
No permissions	---	7

The default *umask* for a standard user is **0002**. The default *umask* for a **root** user is **0022**.

The first digit of the *umask* represents special permissions (sticky bit,). The last three digits of the *umask* represent the permissions that are removed from the user owner (**u**), group owner (**g**), and others (**o**) respectively.

Example 25.3. Applying the umask when creating a file

The following example illustrates how the *umask* with an octal value of **0137** is applied to the file with the base permission of **777**, to create the file with the default permission of **640**.



25.3. DEFAULT FILE PERMISSIONS

The default permissions are set automatically for all newly created files and directories. The value of the default permissions is determined by applying the *umask* to the base permission.

Example 25.4. Default permissions for a directory created by a standard user

When a **standard user** creates a new **directory**, the *umask* is set to **002 (rwxrwxr-x)**, and the base permissions for a directory are set to **777 (rwxrwxrwx)**. This brings the default permissions to **775 (drwxrwxr-x)**.

	Symbolic value	Octal value
Base permission	rwxrwxrwx	777
Umask	rwxrwxr-x	002
Default permission	rwxrwxr-x	775

This means that the directory owner and the group can list the contents of the directory, create, delete, and edit items within the directory, and descend into it. Other users can only list the contents of the directory and descend into it.

Example 25.5. Default permissions for a file created by a standard user

When a **standard user** creates a new **file**, the *umask* is set to **002 (rwxrwxr-x)**, and the base permissions for a file are set to **666 (rw-rw-rw-)**. This brings the default permissions to **664 (-rw-rw-r-)**.

	Symbolic value	Octal value
Base permission	rw-rw-rw-	666
Umask	rw-rw-rw-	002
Default permission	rw-rw-r--	664

This means that the file owner and the group can read and edit the file, while other users can only read the file.

Example 25.6. Default permissions for a directory created by the root user

When a **root user** creates a new **directory**, the *umask* is set to **022 (rwxr-xr-x)**, and the base permissions for a directory are set to **777 (rwxrwxrwx)**. This brings the default permissions to **755 (rwxr-xr-x)**.

	Symbolic value	Octal value
Base permission	rwxrwxrwx	777
Umask	rwxr-xr-x	022
Default permission	rwxr-xr-x	755

This means that the directory owner can list the contents of the directory, create, delete, and edit items within the directory, and descend into it. The group and others can only list the contents of the directory and descend into it.

Example 25.7. Default permissions for a file created by the root user

When a **root user** creates a new **file**, the *umask* is set to **022 (rwxr-xr-x)**, and the base permissions for a file are set to **666 (rw-rw-rw-)**. This brings the default permissions to **644 (-rw-r--r-)**.

	Symbolic value	Octal value
Base permission	rw-rw-rw-	666
Umask	rwxr-xr-x	022
Default permission	rw-r--r--	644

This means that the file owner can read and edit the file, while the group and others can only read the file.

**NOTE**

For security reasons, regular files cannot have execute permissions by default, even if the `umask` is set to **000** (**rw-rw-rw-**). However, directories can be created with execute permissions.

25.4. CHANGING FILE PERMISSIONS USING SYMBOLIC VALUES

You can use the **chmod** utility with symbolic values (a combination letters and signs) to change file permissions for a file or directory.

You can assign the following *permissions*:

- Read (**r**)
- Write (**w**)
- Execute (**x**)

Permissions can be assigned to the following *levels of ownership*:

- User owner (**u**)
- Group owner (**g**)
- Other (**o**)
- All (**a**)

To add or remove permissions you can use the following *signs*:

- **+** to add the permissions on top of the existing permissions
- **-** to remove the permissions from the existing permission
- **=** to remove the existing permissions and explicitly define the new ones

Procedure

- To change the permissions for a file or directory, use:

```
$ chmod <level><operation><permission> file-name
```

Replace **<level>** with the [level of ownership](#) you want to set the permissions for. Replace **<operation>** with one of the [signs](#). Replace **<permission>** with the [permissions](#) you want to assign. Replace *file-name* with the name of the file or directory. For example, to grant everyone the permissions to read, write, and execute (**rw-rw-rw-**) **my-script.sh**, use the **chmod a=rwx my-script.sh** command.

See [Base permissions](#) for more details.

Verification steps

- To see the permissions for a particular file, use:

```
$ ls -l file-name
```


-

Replace *file-name* with the name of the file.

- To see the permissions for a particular directory, use:

```
$ ls -dl directory-name
```

Replace *directory-name* with the name of the directory.

- To see the permissions for all the files within a particular directory, use:

```
$ ls -l directory-name
```

Replace *directory-name* with the name of the directory.

Example 25.8. Changing permissions for files and directories

- To change file permissions for **my-file.txt** from **-rw-rw-r--** to **-rw-----**, use:

1. Display the current permissions for **my-file.txt**:

```
$ ls -l my-file.txt
-rw-rw-r--. 1 username username 0 Feb 24 17:56 my-file.txt
```

2. Remove the permissions to read, write, and execute (**rw**x) the file from group owner (**g**) and others (**o**):

```
$ chmod go= my-file.txt
```

Note that any permission that is not specified after the equals sign (=) is automatically prohibited.

3. Verify that the permissions for **my-file.txt** were set correctly:

```
$ ls -l my-file.txt
-rw-----. 1 username username 0 Feb 24 17:56 my-file.txt
```

- To change file permissions for **my-directory** from **drwxrwx---** to **drwxrwxr-x**, use:

1. Display the current permissions for **my-directory**:

```
$ ls -dl my-directory
drwxrwx---. 2 username username 4096 Feb 24 18:12 my-directory
```

2. Add the read, write, execute (**rw**x) access for all users (**a**):

```
$ chmod o+rx my-directory
```

3. Verify that the permissions for **my-directory** and its content were set correctly:

```
$ ls -dl my-directory
drwxrwxr-x. 2 username username 4096 Feb 24 18:12 my-directory
```

25.5. CHANGING FILE PERMISSIONS USING OCTAL VALUES

You can use the **chmod** utility with octal values (numbers) to change file permissions for a file or directory.

Procedure

- To change the file permissions for an existing file or directory, use:

```
$ chmod octal_value file-name
```

Replace *file-name* with the name of the file or directory. Replace *octal_value* with an octal value. See [Base permissions](#) for more details.

CHAPTER 26. MANAGING THE UMASK

You can use the **umask** utility to display, set, or change the current or default value of the *umask*.

26.1. DISPLAYING THE CURRENT VALUE OF THE UMASK

You can use the **umask** utility to display the current value of the *umask* in symbolic or octal mode.

Procedure

- To display the current value of the *umask* in symbolic mode, use:

```
$ umask -S
```

- To display the current value of the *umask* in the octal mode, use:

```
$ umask
```



NOTE

When displaying the *umask* in octal mode, you may notice it displayed as a four digit number (**0002** or **0022**). The first digit of the *umask* represents a special bit (sticky bit, SGID bit, or SUID bit). If the first digit is set to **0**, the special bit is not set.

26.2. DISPLAYING THE DEFAULT BASH UMASK

There are a number of shells you can use, such as **bash**, **ksh**, **zsh** and **tcsh**. Those shells can behave as login or non-login shells. The login shell is typically invoked by opening a native or a GUI terminal.

To determine whether you are executing a command in a login or a non-login shell, use the **echo \$0** command.

Example 26.1. Determining if you are working in a login or a non-login bash shell

- If the output of the **echo \$0** command returns **bash**, you are executing the command in a non-login shell.

```
$ echo $0
bash
```

The default *umask* for the non-login shell is set in **/etc/bashrc** configuration file.

- If the output of the **echo \$0** command returns **-bash**, you are executing the command in a login shell.

```
# echo $0
-bash
```

The default *umask* for the login shell is set in **/etc/profile** configuration file.

Procedure

- To display the default **bash** *umask* for the non-login shell, use:

```
$ grep umask /etc/bashrc
```

The output returns:

```
# By default, we want umask to get set. This sets it for non-login shell.
umask 002
umask 022
```

- To display the default **bash** *umask* for the login shell, use:

```
$ grep umask /etc/profile
```

The output returns:

```
# By default, we want umask to get set. This sets it for login shell
umask 002
umask 022
```

26.3. SETTING THE UMASK USING SYMBOLIC VALUES

You can use the **umask** utility with symbolic values (a combination letters and signs) to set the *umask* for the current shell session

You can assign the following *permissions*:

- Read (**r**)
- Write (**w**)
- Execute (**x**)

Permissions can be assigned to the following *levels of ownership*:

- User owner (**u**)
- Group owner (**g**)
- Other (**o**)
- All (**a**)

To add or remove permissions you can use the following *signs*:

- **+** to add the permissions on top of the existing permissions
- **-** to remove the permissions from the existing permission
- **=** to remove the existing permissions and explicitly define the new ones

**NOTE**

Any permission that is not specified after the equals sign (=) is automatically prohibited.

Procedure

- To set the *umask* for the current shell session, use:

```
$ umask -S <level><operation><permission>
```

Replace **<level>** with the [level of ownership](#) you want to set the *umask* for. Replace **<operation>** with one of the [signs](#). Replace **<permission>** with the [permissions](#) you want to assign. For example, to set the *umask* to **u=rwx,g=rwx,o=rwx**, use **umask -S a=rwx**.

See [User file creation mode](#) for more details.

**NOTE**

The *umask* is only valid for the current shell session.

26.4. SETTING THE UMASK USING OCTAL VALUES

You can use the **umask** utility with octal values (numbers) to set the *umask* for the current shell session.

Procedure

- To set the *umask* for the current shell session, use:

```
$ umask octal_value
```

Replace *octal_value* with an octal value. See [User file creation mode mask](#) for more details.

**NOTE**

The *umask* is only valid for the current shell session.

26.5. CHANGING THE DEFAULT UMASK FOR THE NON-LOGIN SHELL

You can change the default **bash** *umask* for standard users by modifying the **/etc/bashrc** file.

Prerequisites

- **root** access

Procedure

1. As **root**, open the **/etc/bashrc** file in an editor of your choice.
2. Modify the following sections to set a new default bash *umask*:

```
if [ $UID -gt 199 ] && [ "id -gn" = "id -un" ]; then
    umask 002
```

```
else
    umask 022
fi
```

Replace the default octal value of the *umask* (**002**) with another octal value. See [User file creation mode mask](#) for more details.

3. Save the changes and exit the editor.

26.6. CHANGING THE DEFAULT UMASK FOR THE LOGIN SHELL

You can change the default **bash** *umask* for the **root** user by modifying the `/etc/profile` file.

Prerequisites

- **root** access

Procedure

1. As **root**, open the `/etc/profile` file in an editor of your choice.
2. Modify the following sections to set a new default bash *umask*:

```
if [ $UID -gt 199 ] && [ "/usr/bin/id -gn" = "/usr/bin/id -un" ]; then
    umask 002
else
    umask 022
fi
```

Replace the default octal value of the *umask* (**022**) with another octal value. See [User file creation mode mask](#) for more details.

3. Save the changes and exit the editor.

26.7. CHANGING THE DEFAULT UMASK FOR A SPECIFIC USER

You can change the default *umask* for a specific user by modifying the `.bashrc` for that user.

Procedure

- Append the line that specifies the octal value of the *umask* into the `.bashrc` file for the particular user.

```
$ echo 'umask octal_value' >> /home/username/.bashrc
```

Replace *octal_value* with an octal value and replace *username* with the name of the user. See [User file creation mode mask](#) for more details.

26.8. SETTING DEFAULT UMASK FOR NEWLY CREATED HOME DIRECTORIES

You can change the permissions that specify the *UMASK* for home directories of newly created users by modifying the `/etc/login.defs` file.

Procedure

1. As **root**, open the **/etc/login.defs** file in an editor of your choice.
2. Modify the following section to set a new default *UMASK*:

```
# The permission mask is initialized to this value. If not specified,  
# the permission mask will be initialized to 022.  
UMASK 077
```

Replace the default octal value (**077**) with another octal value. See [User file creation mode mask](#) for more details.

3. Save the changes and exit the editor.

CHAPTER 27. USING DNSTAP IN RHEL

The **dnstap** utility provides an advanced way to monitor and log details of incoming name queries. It records sent messages from the **named** service. This section explains how to record DNS queries using **dnstap**.

27.1. RECORDING DNS QUERIES USING DNSTAP IN RHEL

The network administrators can record the DNS queries to collect the website or IP address information along with the domain health.

Prerequisites

- Upgrade **BIND** packages to version **bind-9.11.26-2** or newer.



WARNING

If you already have a **BIND** version installed and running, adding a new version of **BIND** will overwrite the existing version.

Procedure

Following are the steps to record DNS queries:

1. Edit the **/etc/named.conf** file in the **options** block to enable **dnstap** and target file:

```
options
{
# ...

dnstap { all; }; # Configure filter
dnstap-output file "/var/named/data/dnstap.bin";

# ...
};
# end of options
```

(all | auth | client | forwarder | resolver | update) [(query | response)];

The **dnstap** filter contains multiple definitions delimited by a **;** in the **dnstap {}** block.

Following is the syntax for each rule:

- **auth** – Authoritative zone response or answer.
- **client** – Internal client query or answer.
- **forwarder** – Forwarded query or response from it.
- **resolver** – Iterative resolution query or response.

- **update** - Dynamic zone update requests.
- **all** - Any from the above options.
- **query | response** - If no query or response keyword is specified, both would be recorded. The following example requests **auth** responses only, **client queries** and both queries and responses of dynamic **updates**:

Example:

```
dnstap {auth response; client query; update;};
```

2. Configure the periodic rollout for the active logs.
In the following example, the content of the user-edited script run once per day by **cron**. Number 3 signifies the backup log files limited to that number. Because the file gets removed, it never reaches the **.2** suffix.

Example:

```
sudoedit /etc/cron.daily/dnstap
```

```
#!/bin/sh
```

```
rndc dnstap -roll 3
```

```
mv /var/named/data/dnstap.bin.1 \ /var/log/named/dnstap/dnstap-$(date -l).bin
```

```
# use dnstap-read to analyze saved logs
```

```
sudo chmod a+x /etc/cron.daily/dnstap
```

3. Use the **dnstap-read** utility to handle and analyze the logs in a human-readable format.
In the following example, the detailed dnstap output gets printed in the **YAML** file format.

Example:

```
dnstap-read -y [file-name]
```

CHAPTER 28. MANAGING THE ACCESS CONTROL LIST

Each file and directory can only have one user owner and one group owner at a time. If you want to grant a user permissions to access specific files or directories that belong to a different user or group while keeping other files and directories private, you can utilize Linux Access Control Lists (ACLs).

28.1. DISPLAYING THE CURRENT ACCESS CONTROL LIST

You can use the **getfacl** utility to display the current ACL.

Procedure

- To display the current ACL for a particular file or directory, use:

```
$ getfacl file-name
```

Replace *file-name* with the name of the file or directory.

28.2. SETTING THE ACCESS CONTROL LIST

You can use the **setfacl** utility to set the ACL for a file or directory.

Prerequisites

- **root** access.

Procedure

- To set the ACL for a file or directory, use:

```
# setfacl -m u:username:symbolic_value file-name
```

Replace *username* with the name of the user, *symbolic_value* with a symbolic value, and *file-name* with the name of the file or directory. For more information see the **setfacl** man page.

Example 28.1. Modifying permissions for a group project

The following example describes how to modify permissions for the **group-project** file owned by the **root** user that belongs to the **root** group so that this file is:

- Not executable by anyone.
- The user **andrew** has the **rw-** permissions.
- The user **susan** has the **---** permissions.
- Other users have the **r--** permissions.

Procedure

```
# setfacl -m u:andrew:rw- group-project  
# setfacl -m u:susan:--- group-project
```

Verification steps

- To verify that the user **andrew** has the **rw-** permission, the user **susan** has the **---** permission, and other users have the **r--** permission, use:

```
$ getfacl group-project
```

The output returns:

```
# file: group-project
# owner: root
# group: root
user:andrew:rw-
user:susan:---
group::r--
mask::rw-
other::r--
```

CHAPTER 29. USING THE CHRONY SUITE TO CONFIGURE NTP

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.

Starting with Red Hat Enterprise Linux 8, the **NTP** protocol is implemented by the **chronyd** daemon, available from the repositories in the **chrony** package.

The following sections describe how to use the **chrony** suite to configure NTP.

29.1. 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.

29.2. USING CHRONYC TO CONTROL CHRONYD

This section describes how to control **chronyd** using the **chronyc** command line utility.

Procedure

1. 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>
```

2. To list all of the commands, type **help**.
3. Alternatively, 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**.

29.3. 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](#).

Starting with 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 29.1. Corresponding names of the programs, configuration files and services when migrating from ntp to chrony

ntp name	chrony name
/etc/ntp.conf	/etc/chrony.conf
/etc/ntp/keys	/etc/chrony.keys
ntpd	chronyd
ntpq	chronyc
ntpd.service	chronyd.service
ntp-wait.service	chrony-wait.service

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 +SIGND +ASYNCDNS +SECHASH +IPV6 +DEBUG)
2018-05-18T12:37:43Z Initial frequency -2.630 ppm
2018-05-18T12:37:48Z System clock wrong by 0.003159 seconds (step)
2018-05-18T12:37:48Z chronyd exiting
```

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.

29.3.1. 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**.

CHAPTER 30. USING CHRONY

The following sections describe how to install, start, and stop **chronyd**, and how to check if **chrony** is synchronized. Sections also describe how to manually adjust System Clock.

30.1. MANAGING CHRONY

The following procedure describes how to install, start, stop, and check the status of **chronyd**.

Procedure

1. 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**.

2. 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
```

3. To start **chronyd**, issue the following command as **root**:

```
# systemctl start chronyd
```

To ensure **chronyd** starts automatically at system start, issue the following command as **root**:

```
# systemctl enable chronyd
```

4. 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
```

30.2. CHECKING IF CHRONY IS SYNCHRONIZED

The following procedure describes how to check if **chrony** is synchronized with the use of the **tracking**, **sources**, and **sourcestats** commands.

Procedure

1. 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 **sources** command displays information about the current time sources that **chronyd** is accessing. To check **chrony** sources, issue the following command:

\$ 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 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.

- The **sourcestats** command displays information about the drift rate and offset estimation process for each of the sources currently being examined by **chronyd**. To check **chrony** source statistics, issue the following command:

\$ 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 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.

Additional resources

- **chronyc(1)** man page

30.3. MANUALLY ADJUSTING THE SYSTEM CLOCK

The following procedure describes how to manually adjust the System Clock.

Procedure

1. To step the system clock immediately, bypassing any adjustments in progress by slewing, issue the following command as **root**:

```
# chronyc makestep
```

If the **rtcf** 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.

30.4. 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.

The following procedure describes how to set up **chrony** for a system in an isolated network.

Procedure

1. 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.

2. 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.

30.5. ADDITIONAL RESOURCES

- **chronyc(1)** man page
- **chronyd(8)** man page
- [Frequently Asked Questions](#)

CHAPTER 31. CHRONY WITH HW 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.

The following sections describe how to:

- Verify support for hardware timestamping
- Enable hardware timestamping
- Configure client polling interval
- Enable interleaved mode
- Configure server for large number of clients
- Verify hardware timestamping
- Configure PTP-NTP bridge

31.1. 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 31.1. Verifying support for hardware timestamping on a specific interface

```
# 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)
ptpv2-event        (HWTSTAMP_FILTER_PTP_V2_EVENT)
ptpv2-sync         (HWTSTAMP_FILTER_PTP_V2_SYNC)
ptpv2-delay-req    (HWTSTAMP_FILTER_PTP_V2_DELAY_REQ)

```

31.2. 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 31.2. Enabling hardware timestamping by using the hwtimestamp directive

```

hwtimestamp eth0
hwtimestamp eth1
hwtimestamp *

```

31.3. 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
```

31.4. 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
```

31.5. 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
```

31.6. 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 31.3. 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 31.4. Reporting the transmit, receive timestamping and interleaved mode for each NTP source

```
# chronyc ntpdata
```

Output:

```
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 31.5. Reporting the stability of NTP measurements

```
# chronyc sourcestats
```

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
```

31.7. 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 31.6. Configuring chronyd to provide the system time using the other interface

```
bindaddress 203.0.113.74
hwtimestamp eth1
local stratum 1
```

CHAPTER 32. 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**. The following sections list such settings, and describe ways to achieve them on a system with **chrony**.

32.1. MONITORING BY NTPQ AND NTPDC

chrony 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 32.1. Using the tracking command

```
$ 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 32.2. Using the ntpstat utility

```
$ ntpstat
synchronised to NTP server (10.5.27.10) at stratum 2
time correct to within 80 ms
polling server every 64 s
```

32.2. 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.

32.3. 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.

32.4. 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.

CHAPTER 33. USING SECURE COMMUNICATIONS BETWEEN TWO SYSTEMS WITH OPENSSH

SSH (Secure Shell) is a protocol which provides secure communications between two systems using a client-server architecture and allows users to log in to server host systems remotely. Unlike other remote communication protocols, such as FTP or Telnet, SSH encrypts the login session, which prevents intruders to collect unencrypted passwords from the connection.

Red Hat Enterprise Linux includes the basic **OpenSSH** packages: the general **openssh** package, the **openssh-server** package and the **openssh-clients** package. Note that the **OpenSSH** packages require the **OpenSSL** package **openssl-libs**, which installs several important cryptographic libraries that enable **OpenSSH** to provide encrypted communications.

33.1. SSH AND OPENSSH

SSH (Secure Shell) is a program for logging into a remote machine and executing commands on that machine. The SSH protocol provides secure encrypted communications between two untrusted hosts over an insecure network. You can also forward X11 connections and arbitrary TCP/IP ports over the secure channel.

The SSH protocol mitigates security threats, such as interception of communication between two systems and impersonation of a particular host, when you use it for remote shell login or file copying. This is because the SSH client and server use digital signatures to verify their identities. Additionally, all communication between the client and server systems is encrypted.

A host key authenticates hosts in the SSH protocol. Host keys are cryptographic keys that are generated automatically when **OpenSSH** is first installed, or when the host boots for the first time.

OpenSSH is an implementation of the SSH protocol supported by a number of Linux, UNIX, and similar operating systems. It includes the core files necessary for both the OpenSSH client and server. The OpenSSH suite consists of the following user-space tools:

- **ssh** is a remote login program (SSH client)
- **sshd** is an **OpenSSH** SSH daemon
- **scp** is a secure remote file copy program
- **sftp** is a secure file transfer program
- **ssh-agent** is an authentication agent for caching private keys
- **ssh-add** adds private key identities to **ssh-agent**
- **ssh-keygen** generates, manages, and converts authentication keys for **ssh**
- **ssh-copy-id** is a script that adds local public keys to the **authorized_keys** file on a remote SSH server
- **ssh-keyscan** - gathers SSH public host keys

Two versions of SSH currently exist: version 1, and the newer version 2. The **OpenSSH** suite in RHEL supports only SSH version 2, which has an enhanced key-exchange algorithm not vulnerable to known exploits in version 1.

OpenSSH, as one of the RHEL core cryptographic subsystems uses system-wide crypto policies. This

ensures that weak cipher suites and cryptographic algorithms are disabled in the default configuration. To adjust the policy, the administrator must either use the **update-crypto-policies** command to make settings stricter or looser or manually opt-out of the system-wide crypto policies.

The **OpenSSH** suite uses two different sets of configuration files: those for client programs (that is, **ssh**, **scp**, and **sftp**), and those for the server (the **sshd** daemon). System-wide SSH configuration information is stored in the `/etc/ssh/` directory. User-specific SSH configuration information is stored in `~/.ssh/` in the user's home directory. For a detailed list of OpenSSH configuration files, see the **FILES** section in the **sshd(8)** man page.

Additional resources

- Man pages listed by using the **man -k ssh** command.
- [Using system-wide cryptographic policies](#).

33.2. CONFIGURING AND STARTING AN OPENSSSH SERVER

Use the following procedure for a basic configuration that might be required for your environment and for starting an **OpenSSH** server. Note that after the default RHEL installation, the **sshd** daemon is already started and server host keys are automatically created.

Prerequisites

- The **openssh-server** package is installed.

Procedure

1. Start the **sshd** daemon in the current session and set it to start automatically at boot time:

```
# systemctl start sshd
# systemctl enable sshd
```

2. To specify different addresses than the default **0.0.0.0** (IPv4) or **::** (IPv6) for the **ListenAddress** directive in the `/etc/ssh/sshd_config` configuration file and to use a slower dynamic network configuration, add the dependency on the **network-online.target** target unit to the **sshd.service** unit file. To achieve this, create the `/etc/systemd/system/sshd.service.d/local.conf` file with the following content:

```
[Unit]
Wants=network-online.target
After=network-online.target
```

3. Review if **OpenSSH** server settings in the `/etc/ssh/sshd_config` configuration file meet the requirements of your scenario.
4. Optionally, change the welcome message that your **OpenSSH** server displays before a client authenticates by editing the `/etc/issue` file, for example:

```
Welcome to ssh-server.example.com
Warning: By accessing this server, you agree to the referenced terms and conditions.
```

Ensure that the **Banner** option is not commented out in `/etc/ssh/sshd_config` and its value contains `/etc/issue`:

```
# less /etc/ssh/sshd_config | grep Banner
Banner /etc/issue
```

Note that to change the message displayed after a successful login you have to edit the **/etc/motd** file on the server. See the **pam_motd** man page for more information.

5. Reload the **systemd** configuration and restart **sshd** to apply the changes:

```
# systemctl daemon-reload
# systemctl restart sshd
```

Verification

1. Check that the **sshd** daemon is running:

```
# systemctl status sshd
● sshd.service - OpenSSH server daemon
   Loaded: loaded (/usr/lib/systemd/system/sshd.service; enabled; vendor preset: enabled)
   Active: active (running) since Mon 2019-11-18 14:59:58 CET; 6min ago
     Docs: man:sshd(8)
           man:sshd_config(5)
  Main PID: 1149 (sshd)
    Tasks: 1 (limit: 11491)
   Memory: 1.9M
    CGroup: /system.slice/sshd.service
            └─1149 /usr/sbin/sshd -D -oCiphers=aes128-ctr,aes256-ctr,aes128-cbc,aes256-cbc -
              oMACs=hmac-sha2-256,>

Nov 18 14:59:58 ssh-server-example.com systemd[1]: Starting OpenSSH server daemon...
Nov 18 14:59:58 ssh-server-example.com sshd[1149]: Server listening on 0.0.0.0 port 22.
Nov 18 14:59:58 ssh-server-example.com sshd[1149]: Server listening on :: port 22.
Nov 18 14:59:58 ssh-server-example.com systemd[1]: Started OpenSSH server daemon.
```

2. Connect to the SSH server with an SSH client.

```
# ssh user@ssh-server-example.com
ECDSA key fingerprint is SHA256:dXbaS0RG/UzITTKu8GtXSz0S1++IPegSy31v3L/FAEc.
Are you sure you want to continue connecting (yes/no/[fingerprint])? yes
Warning: Permanently added 'ssh-server-example.com' (ECDSA) to the list of known hosts.

user@ssh-server-example.com's password:
```

Additional resources

- **sshd(8)** and **sshd_config(5)** man pages.

33.3. SETTING AN OPENSSH SERVER FOR KEY-BASED AUTHENTICATION

To improve system security, enforce key-based authentication by disabling password authentication on your OpenSSH server.

Prerequisites

- The **openssh-server** package is installed.
- The **sshd** daemon is running on the server.

Procedure

1. Open the **/etc/ssh/sshd_config** configuration in a text editor, for example:

```
# vi /etc/ssh/sshd_config
```

2. Change the **PasswordAuthentication** option to **no**:

```
PasswordAuthentication no
```

On a system other than a new default installation, check that **PubkeyAuthentication no** has not been set and the **ChallengeResponseAuthentication** directive is set to **no**. If you are connected remotely, not using console or out-of-band access, test the key-based login process before disabling password authentication.

3. To use key-based authentication with NFS-mounted home directories, enable the **use_nfs_home_dirs** SELinux boolean:

```
# setsebool -P use_nfs_home_dirs 1
```

4. Reload the **sshd** daemon to apply the changes:

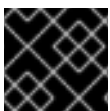
```
# systemctl reload sshd
```

Additional resources

- **sshd(8)**, **sshd_config(5)**, and **setsebool(8)** man pages.

33.4. GENERATING SSH KEY PAIRS

Use this procedure to generate an SSH key pair on a local system and to copy the generated public key to an **OpenSSH** server. If the server is configured accordingly, you can log in to the **OpenSSH** server without providing any password.



IMPORTANT

If you complete the following steps as **root**, only **root** is able to use the keys.

Procedure

1. To generate an ECDSA key pair for version 2 of the SSH protocol:

```
$ ssh-keygen -t ecdsa
Generating public/private ecdsa key pair.
Enter file in which to save the key (/home/joesec/.ssh/id_ecdsa):
Enter passphrase (empty for no passphrase):
Enter same passphrase again:
Your identification has been saved in /home/joesec/.ssh/id_ecdsa.
Your public key has been saved in /home/joesec/.ssh/id_ecdsa.pub.
```

```

The key fingerprint is:
SHA256:Q/x+qms4j7PCQ0qFd09iZEFHA+SqwBKRNau72oZfaCI
joesec@localhost.example.com
The key's randomart image is:
+---[ECDSA 256]---+
|.00..0=++      |
|.. 0 .00 .     |
|. .. 0. 0      |
|...0.+...      |
|0.00.0 +S .    |
|.=.+ .0        |
|E.*+ . . .     |
|.=..+ +.. 0    |
| . 00*+0.      |
+----[SHA256]-----+

```

You can also generate an RSA key pair by using the **-t rsa** option with the **ssh-keygen** command or an Ed25519 key pair by entering the **ssh-keygen -t ed25519** command.

2. To copy the public key to a remote machine:

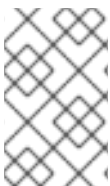
```

$ ssh-copy-id joesec@ssh-server-example.com
/usr/bin/ssh-copy-id: INFO: attempting to log in with the new key(s), to filter out any that are
already installed
joesec@ssh-server-example.com's password:
...
Number of key(s) added: 1

Now try logging into the machine, with: "ssh 'joesec@ssh-server-example.com'" and check to
make sure that only the key(s) you wanted were added.

```

If you do not use the **ssh-agent** program in your session, the previous command copies the most recently modified **~/.ssh/id*.pub** public key if it is not yet installed. To specify another public-key file or to prioritize keys in files over keys cached in memory by **ssh-agent**, use the **ssh-copy-id** command with the **-i** option.



NOTE

If you reinstall your system and want to keep previously generated key pairs, back up the **~/.ssh/** directory. After reinstalling, copy it back to your home directory. You can do this for all users on your system, including **root**.

Verification

1. Log in to the OpenSSH server without providing any password:

```

$ ssh joesec@ssh-server-example.com
Welcome message.
...
Last login: Mon Nov 18 18:28:42 2019 from ::1

```

Additional resources

- **ssh-keygen(1)** and **ssh-copy-id(1)** man pages.

33.5. USING SSH KEYS STORED ON A SMART CARD

Red Hat Enterprise Linux enables you to use RSA and ECDSA keys stored on a smart card on OpenSSH clients. Use this procedure to enable authentication using a smart card instead of using a password.

Prerequisites

- On the client side, the **opensc** package is installed and the **pcscd** service is running.

Procedure

- List all keys provided by the OpenSC PKCS #11 module including their PKCS #11 URIs and save the output to the *keys.pub* file:

```
$ ssh-keygen -D pkcs11: > keys.pub
$ ssh-keygen -D pkcs11:
ssh-rsa AAAAB3NzaC1yc2E...KKZMzcQZzx
pkcs11:id=%02;object=SIGN%20pubkey;token=SSH%20key;manufacturer=piv_II?module-
path=/usr/lib64/pkcs11/opensc-pkcs11.so
ecdsa-sha2-nistp256 AAA...J0hkYnnsM=
pkcs11:id=%01;object=PIV%20AUTH%20pubkey;token=SSH%20key;manufacturer=piv_II?
module-path=/usr/lib64/pkcs11/opensc-pkcs11.so
```

- To enable authentication using a smart card on a remote server (*example.com*), transfer the public key to the remote server. Use the **ssh-copy-id** command with *keys.pub* created in the previous step:

```
$ ssh-copy-id -f -i keys.pub username@example.com
```

- To connect to *example.com* using the ECDSA key from the output of the **ssh-keygen -D** command in step 1, you can use just a subset of the URI, which uniquely references your key, for example:

```
$ ssh -i "pkcs11:id=%01?module-path=/usr/lib64/pkcs11/opensc-pkcs11.so" example.com
Enter PIN for 'SSH key':
[example.com] $
```

- You can use the same URI string in the *~/.ssh/config* file to make the configuration permanent:

```
$ cat ~/.ssh/config
IdentityFile "pkcs11:id=%01?module-path=/usr/lib64/pkcs11/opensc-pkcs11.so"
$ ssh example.com
Enter PIN for 'SSH key':
[example.com] $
```

Because OpenSSH uses the **p11-kit-proxy** wrapper and the OpenSC PKCS #11 module is registered to PKCS#11 Kit, you can simplify the previous commands:

```
$ ssh -i "pkcs11:id=%01" example.com
Enter PIN for 'SSH key':
[example.com] $
```

If you skip the **id=** part of a PKCS #11 URI, OpenSSH loads all keys that are available in the proxy module. This can reduce the amount of typing required:

```
$ ssh -i pkcs11: example.com
Enter PIN for 'SSH key':
[example.com] $
```

Additional resources

- [Fedora 28: Better smart card support in OpenSSH](#) .
- **p11-kit(8)**, **opensc.conf(5)**, **pcscd(8)**, **ssh(1)**, and **ssh-keygen(1)** man pages.

33.6. MAKING OPENSSH MORE SECURE

The following tips help you to increase security when using OpenSSH. Note that changes in the **/etc/ssh/sshd_config** OpenSSH configuration file require reloading the **sshd** daemon to take effect:

```
# systemctl reload sshd
```



IMPORTANT

The majority of security hardening configuration changes reduce compatibility with clients that do not support up-to-date algorithms or cipher suites.

Disabling insecure connection protocols

- To make SSH truly effective, prevent the use of insecure connection protocols that are replaced by the **OpenSSH** suite. Otherwise, a user's password might be protected using SSH for one session only to be captured later when logging in using Telnet. For this reason, consider disabling insecure protocols, such as telnet, rsh, rlogin, and ftp.

Enabling key-based authentication and disabling password-based authentication

- Disabling passwords for authentication and allowing only key pairs reduces the attack surface and it also might save users' time. On clients, generate key pairs using the **ssh-keygen** tool and use the **ssh-copy-id** utility to copy public keys from clients on the **OpenSSH** server. To disable password-based authentication on your OpenSSH server, edit **/etc/ssh/sshd_config** and change the **PasswordAuthentication** option to **no**:

```
PasswordAuthentication no
```

Key types

- Although the **ssh-keygen** command generates a pair of RSA keys by default, you can instruct it to generate ECDSA or Ed25519 keys by using the **-t** option. The ECDSA (Elliptic Curve Digital Signature Algorithm) offers better performance than RSA at the equivalent symmetric key strength. It also generates shorter keys. The Ed25519 public-key algorithm is an implementation of twisted Edwards curves that is more secure and also faster than RSA, DSA, and ECDSA. OpenSSH creates RSA, ECDSA, and Ed25519 server host keys automatically if they are missing. To configure the host key creation in RHEL, use the **sshd-keygen@.service** instantiated service. For example, to disable the automatic creation of the RSA key type:

■

```
# systemctl mask sshd-keygen@rsa.service
```

- To exclude particular key types for SSH connections, comment out the relevant lines in `/etc/ssh/ssh_config`, and reload the **sshd** service. For example, to allow only Ed25519 host keys:

```
# HostKey /etc/ssh/ssh_host_rsa_key
# HostKey /etc/ssh/ssh_host_ecdsa_key
HostKey /etc/ssh/ssh_host_ed25519_key
```

Non-default port

- By default, the **sshd** daemon listens on TCP port 22. Changing the port reduces the exposure of the system to attacks based on automated network scanning and thus increase security through obscurity. You can specify the port using the **Port** directive in the `/etc/ssh/ssh_config` configuration file.
You also have to update the default SELinux policy to allow the use of a non-default port. To do so, use the **semanage** tool from the **policycoreutils-python-utils** package:

```
# semanage port -a -t ssh_port_t -p tcp port_number
```

Furthermore, update **firewalld** configuration:

```
# firewall-cmd --add-port port_number/tcp
# firewall-cmd --runtime-to-permanent
```

In the previous commands, replace *port_number* with the new port number specified using the **Port** directive.

No root login

- If your particular use case does not require the possibility of logging in as the root user, you should consider setting the **PermitRootLogin** configuration directive to **no** in the `/etc/ssh/ssh_config` file. By disabling the possibility of logging in as the root user, the administrator can audit which users run what privileged commands after they log in as regular users and then gain root rights.
Alternatively, set **PermitRootLogin** to **prohibit-password**:

```
PermitRootLogin prohibit-password
```

This enforces the use of key-based authentication instead of the use of passwords for logging in as root and reduces risks by preventing brute-force attacks.

Using the X Security extension

- The X server in Red Hat Enterprise Linux clients does not provide the X Security extension. Therefore, clients cannot request another security layer when connecting to untrusted SSH servers with X11 forwarding. Most applications are not able to run with this extension enabled anyway.
By default, the **ForwardX11Trusted** option in the `/etc/ssh/ssh_config.d/05-redhat.conf` file is set to **yes**, and there is no difference between the **ssh -X remote_machine** (untrusted host) and **ssh -Y remote_machine** (trusted host) command.

If your scenario does not require the X11 forwarding feature at all, set the **X11Forwarding** directive in the `/etc/ssh/sshd_config` configuration file to **no**.

Restricting access to specific users, groups, or domains

- The **AllowUsers** and **AllowGroups** directives in the `/etc/ssh/sshd_config` configuration file server enable you to permit only certain users, domains, or groups to connect to your OpenSSH server. You can combine **AllowUsers** and **AllowGroups** to restrict access more precisely, for example:

```
AllowUsers *@192.168.1.*;*@10.0.0.*;!*@192.168.1.2
AllowGroups example-group
```

The previous configuration lines accept connections from all users from systems in 192.168.1.* and 10.0.0.* subnets except from the system with the 192.168.1.2 address. All users must be in the **example-group** group. The OpenSSH server denies all other connections.

Note that using allowlists (directives starting with Allow) is more secure than using blocklists (options starting with Deny) because allowlists block also new unauthorized users or groups.

Changing system-wide cryptographic policies

- **OpenSSH** uses RHEL system-wide cryptographic policies, and the default system-wide cryptographic policy level offers secure settings for current threat models. To make your cryptographic settings more strict, change the current policy level:

```
# update-crypto-policies --set FUTURE
Setting system policy to FUTURE
```

- To opt-out of the system-wide crypto policies for your **OpenSSH** server, uncomment the line with the **CRYPTO_POLICY=** variable in the `/etc/sysconfig/sshd` file. After this change, values that you specify in the **Ciphers**, **MACs**, **KexAlgorithms**, and **GSSAPIKexAlgorithms** sections in the `/etc/ssh/sshd_config` file are not overridden. Note that this task requires deep expertise in configuring cryptographic options.
- See [Using system-wide cryptographic policies](#) in the [RHEL Security hardening](#) title for more information.

Additional resources

- **sshd_config(5)**, **ssh-keygen(1)**, **crypto-policies(7)**, and **update-crypto-policies(8)** man pages.

33.7. CONNECTING TO A REMOTE SERVER USING AN SSH JUMP HOST

Use this procedure for connecting your local system to a remote server through an intermediary server, also called jump host.

Prerequisites

- A jump host accepts SSH connections from your local system.
- A remote server accepts SSH connections only from the jump host.

Procedure

1. Define the jump host by editing the `~/.ssh/config` file on your local system, for example:

```
Host jump-server1
  HostName jump1.example.com
```

- The **Host** parameter defines a name or alias for the host you can use in **ssh** commands. The value can match the real host name, but can also be any string.
 - The **HostName** parameter sets the actual host name or IP address of the jump host.
2. Add the remote server jump configuration with the **ProxyJump** directive to `~/.ssh/config` file on your local system, for example:

```
Host remote-server
  HostName remote1.example.com
  ProxyJump jump-server1
```

3. Use your local system to connect to the remote server through the jump server:

```
$ ssh remote-server
```

The previous command is equivalent to the **ssh -J jump-server1 remote-server** command if you omit the configuration steps 1 and 2.

NOTE

You can specify more jump servers and you can also skip adding host definitions to the configurations file when you provide their complete host names, for example:

```
$ ssh -J jump1.example.com,jump2.example.com,jump3.example.com
remote1.example.com
```

Change the host name-only notation in the previous command if the user names or SSH ports on the jump servers differ from the names and ports on the remote server, for example:

```
$ ssh -J
johndoe@jump1.example.com:75,johndoe@jump2.example.com:75,johndoe@jump3.e
xample.com:75 joesec@remote1.example.com:220
```

Additional resources

- **ssh_config(5)** and **ssh(1)** man pages.

33.8. CONNECTING TO REMOTE MACHINES WITH SSH KEYS USING SSH-AGENT

To avoid entering a passphrase each time you initiate an SSH connection, you can use the **ssh-agent** utility to cache the private SSH key. The private key and the passphrase remain secure.

Prerequisites

- You have a remote host with SSH daemon running and reachable through the network.
- You know the IP address or hostname and credentials to log in to the remote host.
- You have generated an SSH key pair with a passphrase and transferred the public key to the remote machine.

For more information, see [Generating SSH key pairs](#).

Procedure

1. Optional: Verify you can use the key to authenticate to the remote host:

- a. Connect to the remote host using SSH:

```
$ ssh example.user1@198.51.100.1 hostname
```

- b. Enter the passphrase you set while creating the key to grant access to the private key.

```
$ ssh example.user1@198.51.100.1 hostname
host.example.com
```

2. Start the **ssh-agent**.

```
$ eval $(ssh-agent)
Agent pid 20062
```

3. Add the key to **ssh-agent**.

```
$ ssh-add ~/.ssh/id_rsa
Enter passphrase for ~/.ssh/id_rsa:
Identity added: ~/.ssh/id_rsa (example.user0@198.51.100.12)
```

Verification

- Optional: Log in to the host machine using SSH.

```
$ ssh example.user1@198.51.100.1

Last login: Mon Sep 14 12:56:37 2020
```

Note that you did not have to enter the passphrase.

33.9. ADDITIONAL RESOURCES

- **sshd(8)**, **ssh(1)**, **scp(1)**, **sftp(1)**, **ssh-keygen(1)**, **ssh-copy-id(1)**, **ssh_config(5)**, **sshd_config(5)**, **update-crypto-policies(8)**, and **crypto-policies(7)** man pages.
- [OpenSSH Home Page](#).
- [Configuring SELinux for applications and services with non-standard configurations](#) .

- [Controlling network traffic using firewalld](#) .

CHAPTER 34. CONFIGURING A REMOTE LOGGING SOLUTION

To ensure that logs from various machines in your environment are recorded centrally on a logging server, you can configure the **Rsyslog** application to record logs that fit specific criteria from the client system to the server.

34.1. THE RSYSLOG LOGGING SERVICE

The Rsyslog application, in combination with the **systemd-journald** service, provides local and remote logging support in Red Hat Enterprise Linux. The **rsyslogd** daemon continuously reads **syslog** messages received by the **systemd-journald** service from the journal. **rsyslogd** then filters and processes these **syslog** events and records them to **rsyslog** log files or forwards them to other services according to its configuration.

The **rsyslogd** daemon also provides extended filtering, encryption protected relaying of messages, input and output modules, and support for transportation using the TCP and UDP protocols.

In **/etc/rsyslog.conf**, which is the main configuration file for **rsyslog**, you can specify the rules according to which **rsyslogd** handles the messages. Generally, you can classify messages by their source and topic (facility) and urgency (priority), and then assign an action that should be performed when a message fits these criteria.

In **/etc/rsyslog.conf**, you can also see a list of log files maintained by **rsyslogd**. Most log files are located in the **/var/log/** directory. Some applications, such as **httpd** and **samba**, store their log files in a subdirectory within **/var/log/**.

Additional resources

- The **rsyslogd(8)** and **rsyslog.conf(5)** man pages
- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>

34.2. INSTALLING RSYSLOG DOCUMENTATION

The Rsyslog application has extensive documentation that is available at <https://www.rsyslog.com/doc/>, but you can also install the **rsyslog-doc** documentation package locally by following this procedure.

Prerequisites

- You have activated the **AppStream** repository on your system.
- You are authorized to install new packages using **sudo**.

Procedure

- Install the **rsyslog-doc** package:

```
$ sudo yum install rsyslog-doc
```

Verification

- Open the <file:///usr/share/doc/rsyslog/html/index.html> file in a browser of your choice, for example:

```
$ firefox file:///usr/share/doc/rsyslog/html/index.html
```

34.3. CONFIGURING A SERVER FOR REMOTE LOGGING OVER TCP

The Rsyslog application enables you to both run a logging server and configure individual systems to send their log files to the logging server. To use remote logging through TCP, configure both the server and the client. The server collects and analyzes the logs sent by one or more client systems.

With the Rsyslog application, you can maintain a centralized logging system where log messages are forwarded to a server over the network. To avoid message loss when the server is not available, you can configure an action queue for the forwarding action. This way, messages that failed to be sent are stored locally until the server is reachable again. Note that such queues cannot be configured for connections using the UDP protocol.

The **omfwd** plug-in provides forwarding over UDP or TCP. The default protocol is UDP. Because the plug-in is built in, it does not have to be loaded.

By default, **rsyslog** uses TCP on port **514**.

Prerequisites

- **rsyslog** is installed on the server system
- You are logged in as root on the server

Procedure

1. Optional: To use a different port for **rsyslog** traffic, add the **syslogd_port_t** SELinux type to port. For example, enable port **30514**:

```
# semanage port -a -t syslogd_port_t -p tcp 30514
```

2. Optional: To use a different port for **rsyslog** traffic, configure **firewalld** to allow incoming **rsyslog** traffic on that port. For example, allow TCP traffic on port **30514** in zone **zone**:

```
# firewall-cmd --zone=zone --permanent --add-port=30514/tcp
success
```

3. Create a new file in the **/etc/rsyslog.d/** directory named, for example, **remotelog.conf**, and insert the following content:

```
# Define templates before the rules that use them
### Per-Host Templates for Remote Systems ###
template(name="TmplAuthpriv" type="list") {
    constant(value="/var/log/remote/auth/")
    property(name="hostname")
    constant(value="")
    property(name="programname" SecurePath="replace")
    constant(value=".log")
}
```

```

template(name="TplMsg" type="list") {
    constant(value="/var/log/remote/msg/")
    property(name="hostname")
    constant(value="")
    property(name="programname" SecurePath="replace")
    constant(value=".log")
}

# Provides TCP syslog reception
module(load="imtcp")
# Adding this ruleset to process remote messages
ruleset(name="remote1"){
    authpriv.* action(type="omfile" DynaFile="TplAuthpriv")
    *.info;mail.none;authpriv.none;cron.none
    action(type="omfile" DynaFile="TplMsg")
}

input(type="imtcp" port="30514" ruleset="remote1")

```

4. Save the changes to the **/etc/rsyslog.d/remotelog.conf** file.
5. Make sure the **rsyslog** service is running and enabled on the logging server:

```
# systemctl status rsyslog
```

6. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

7. Optional: If **rsyslog** is not enabled, ensure the **rsyslog** service starts automatically after reboot:

```
# systemctl enable rsyslog
```

Your log server is now configured to receive and store log files from the other systems in your environment.

Verification

- Test the syntax of the **/etc/rsyslog.conf** file:

```

# rsyslogd -N 1
rsyslogd: version 8.1911.0-2.el8, config validation run (level 1), master config
/etc/rsyslog.conf
rsyslogd: End of config validation run. Bye.

```

Additional resources

- **rsyslogd(8)** man page.
- **rsyslog.conf(5)** man page.
- **semanage(8)** man page.
- **firewall-cmd(1)** man page.

- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>.

34.4. CONFIGURING REMOTE LOGGING TO A SERVER OVER TCP

Follow this procedure to configure a system for forwarding log messages to a server over the TCP protocol. The **omfwd** plug-in provides forwarding over UDP or TCP. The default protocol is UDP. Because the plug-in is built in, you do not have to load it.

Prerequisites

- The **rsyslog** package is installed on the client systems that should report to the server.
- You have configured the server for remote logging.
- The specified port is permitted in SELinux and open in firewall.

Procedure

1. Create a new file in the **/etc/rsyslog.d/** directory named, for example, **remotelog.conf**, and insert the following content:

```
*.* action(type="omfwd"
        queue.type="linkedlist"
        queue.filename="example_fwd"
        action.resumeRetryCount="-1"
        queue.saveOnShutdown="on"
        target="example.com" port="30514" protocol="tcp"
    )
```

Where:

- **queue.type="linkedlist"** enables a LinkedList in-memory queue,
- **queue.filename** defines a disk storage. The backup files are created with the **example_fwd** prefix in the working directory specified by the preceding global **workDirectory** directive,
- the **action.resumeRetryCount -1** setting prevents **rsyslog** from dropping messages when retrying to connect if server is not responding,
- enabled **queue.saveOnShutdown="on"** saves in-memory data if **rsyslog** shuts down,
- the last line forwards all received messages to the logging server, port specification is optional.

With this configuration, **rsyslog** sends messages to the server but keeps messages in memory if the remote server is not reachable. A file on disk is created only if **rsyslog** runs out of the configured memory queue space or needs to shut down, which benefits the system performance.

2. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

Verification

To verify that the client system sends messages to the server, follow these steps:

1. On the client system, send a test message:

```
# logger test
```

2. On the server system, view the `/var/log/messages` log, for example:

```
# cat /var/log/remote/msg/hostname/root.log
Feb 25 03:53:17 hostname root[6064]: test
```

Where *hostname* is the host name of the client system. Note that the log contains the user name of the user that entered the **logger** command, in this case **root**.

Additional resources

- The **rsyslogd(8)** man page.
- The **rsyslog.conf(5)** man page.
- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>.

34.5. CONFIGURING A SERVER FOR RECEIVING REMOTE LOGGING INFORMATION OVER UDP

The **Rsyslog** application enables you to configure a system to receive logging information from remote systems. To use remote logging through UDP, configure both the server and the client. The receiving server collects and analyzes the logs sent by one or more client systems. By default, **rsyslog** uses UDP on port **514** to receive log information from remote systems.

Follow this procedure to configure a server for collecting and analyzing logs sent by one or more client systems over the UDP protocol.

Prerequisites

- The **rsyslog** utility is installed.

Procedure

1. Optional: To use a different port for **rsyslog** traffic than the default port **514**:
 - a. Add the **syslogd_port_t** SELinux type to the SELinux policy configuration, replacing **portno** with the port number you want **rsyslog** to use:

```
# semanage port -a -t syslogd_port_t -p udp portno
```

- b. Configure **firewalld** to allow incoming **rsyslog** traffic, replacing **portno** with the port number and **zone** with the zone you want **rsyslog** to use:

```
# firewall-cmd --zone=zone --permanent --add-port=portno/udp
success
```

- c. Reload the firewall rules:

```
# firewall-cmd --reload
```

2. Create a new **.conf** file in the **/etc/rsyslog.d/** directory, for example, **remotelogserv.conf**, and insert the following content:

```
# Define templates before the rules that use them
### Per-Host Templates for Remote Systems ###
template(name="TmplAuthpriv" type="list") {
    constant(value="/var/log/remote/auth/")
    property(name="hostname")
    constant(value="/")
    property(name="programname" SecurePath="replace")
    constant(value=".log")
}

template(name="TmplMsg" type="list") {
    constant(value="/var/log/remote/msg/")
    property(name="hostname")
    constant(value="/")
    property(name="programname" SecurePath="replace")
    constant(value=".log")
}

# Provides UDP syslog reception
module(load="imudp")

# This ruleset processes remote messages
ruleset(name="remote1"){
    authpriv.* action(type="omfile" DynaFile="TmplAuthpriv")
    *.info;mail.none;authpriv.none;cron.none
    action(type="omfile" DynaFile="TmplMsg")
}

input(type="imudp" port="514" ruleset="remote1")
```

Where **514** is the port number **rsyslog** uses by default. You can specify a different port instead.

3. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

4. Optional: If **rsyslog** is not enabled, ensure the **rsyslog** service starts automatically after reboot:

```
# systemctl enable rsyslog
```

Verification

1. Verify the syntax of the **/etc/rsyslog.conf** file and all **.conf** files in the **/etc/rsyslog.d/** directory:

```
# rsyslogd -N 1
rsyslogd: version 8.1911.0-2.el8, config validation run (level 1), master config
/etc/rsyslog.conf
```

rsyslogd: End of config validation run. Bye.

Additional resources

- **rsyslogd(8)** man page.
- **rsyslog.conf(5)** man page.
- **semanage(8)** man page.
- **firewall-cmd(1)** man page.
- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>.

34.6. CONFIGURING REMOTE LOGGING TO A SERVER OVER UDP

Follow this procedure to configure a system for forwarding log messages to a server over the UDP protocol. The **omfwd** plug-in provides forwarding over UDP or TCP. The default protocol is UDP. Because the plug-in is built in, you do not have to load it.

Prerequisites

- The **rsyslog** package is installed on the client systems that should report to the server.
- You have configured the server for remote logging as described in [Configuring a server for receiving remote logging information over UDP](#).

Procedure

1. Create a new **.conf** file in the **/etc/rsyslog.d/** directory, for example, **remotelogcli.conf**, and insert the following content:

```
*.* action(type="omfwd"
    queue.type="linkedlist"
    queue.filename="example_fwd"
    action.resumeRetryCount="-1"
    queue.saveOnShutdown="on"
    target="example.com" port="portno" protocol="udp"
)
```

Where:

- **queue.type="linkedlist"** enables a LinkedList in-memory queue.
- **queue.filename** defines a disk storage. The backup files are created with the **example_fwd** prefix in the working directory specified by the preceding global **workDirectory** directive.
- The **action.resumeRetryCount -1** setting prevents **rsyslog** from dropping messages when retrying to connect if the server is not responding.
- **enabled queue.saveOnShutdown="on"** saves in-memory data if **rsyslog** shuts down.
- **portno** is the port number you want **rsyslog** to use. The default value is **514**.

- The last line forwards all received messages to the logging server, port specification is optional.
With this configuration, **rsyslog** sends messages to the server but keeps messages in memory if the remote server is not reachable. A file on disk is created only if **rsyslog** runs out of the configured memory queue space or needs to shut down, which benefits the system performance.

2. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

3. Optional: If **rsyslog** is not enabled, ensure the **rsyslog** service starts automatically after reboot:

```
# systemctl enable rsyslog
```

Verification

To verify that the client system sends messages to the server, follow these steps:

1. On the client system, send a test message:

```
# logger test
```

2. On the server system, view the **/var/log/remote/msg/hostname/root.log** log, for example:

```
# cat /var/log/remote/msg/hostname/root.log
Feb 25 03:53:17 hostname root[6064]: test
```

Where **hostname** is the host name of the client system. Note that the log contains the user name of the user that entered the logger command, in this case **root**.

Additional resources

- The **rsyslogd(8)** man page.
- The **rsyslog.conf(5)** man page.
- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>.

34.7. CONFIGURING RELIABLE REMOTE LOGGING

With the Reliable Event Logging Protocol (RELP), you can send and receive **syslog** messages over TCP with a much reduced risk of message loss. RELP provides reliable delivery of event messages, which makes it useful in environments where message loss is not acceptable. To use RELP, configure the **imrelp** input module, which runs on the server and receives the logs, and the **omrelp** output module, which runs on the client and sends logs to the logging server.

Prerequisites

- You have installed the **rsyslog**, **librelp**, and **rsyslog-relp** packages on the server and the client systems.
- The specified port is permitted in SELinux and open in the firewall.

Procedure

1. Configure the client system for reliable remote logging:

- a. On the client system, create a new **.conf** file in the **/etc/rsyslog.d/** directory named, for example, **relpli.conf**, and insert the following content:

```
module(load="omrelp")
*. * action(type="omrelp" target="target_IP" port="target_port")
```

Where:

- ***target_IP*** is the IP address of the logging server.
- ***target_port*** is the port of the logging server.

- b. Save the changes to the **/etc/rsyslog.d/relpserv.conf** file.
- c. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

- d. Optional: If **rsyslog** is not enabled, ensure the **rsyslog** service starts automatically after reboot:

```
# systemctl enable rsyslog
```

2. Configure the server system for reliable remote logging:

- a. On the server system, create a new **.conf** file in the **/etc/rsyslog.d/** directory named, for example, **relpserv.conf**, and insert the following content:

```
ruleset(name="relp"){
*. * action(type="omfile" file="log_path")
}

module(load="imrelp")
input(type="imrelp" port="target_port" ruleset="relp")
```

Where:

- ***log_path*** specifies the path for storing messages.
- ***target_port*** is the port of the logging server. Use the same value as in the client configuration file.

- b. Save the changes to the **/etc/rsyslog.d/relpserv.conf** file.
- c. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

- d. Optional: If **rsyslog** is not enabled, ensure the **rsyslog** service starts automatically after reboot:

■

```
# systemctl enable rsyslog
```

Verification

To verify that the client system sends messages to the server, follow these steps:

1. On the client system, send a test message:

```
# logger test
```

2. On the server system, view the log at the specified **log_path**, for example:

```
# cat /var/log/remote/msg/hostname/root.log
Feb 25 03:53:17 hostname root[6064]: test
```

Where **hostname** is the host name of the client system. Note that the log contains the user name of the user that entered the logger command, in this case **root**.

Additional resources

- The **rsyslogd(8)** man page.
- The **rsyslog.conf(5)** man page.
- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>.

34.8. SUPPORTED RSYSLOG MODULES

To expand the functionality of the **Rsyslog** utility, you can use specific additional modules. Modules provide additional inputs (Input Modules), outputs (Output Modules), and other specific functionalities. A module may also provide additional configuration directives that become available after you load that module.

You can list the input and output modules installed on your system with the following command:

```
# ls /usr/lib64/rsyslog/{i,o}m*
```

You can view the list of all available **rsyslog** modules at the following page from documentation installed from the **rsyslog-doc** package:

file:///usr/share/doc/rsyslog/html/configuration/modules/idx_output.html.

34.9. ADDITIONAL RESOURCES

- Documentation installed with the **rsyslog-doc** package at <file:///usr/share/doc/rsyslog/html/index.html>.
- The **rsyslog.conf(5)** and **rsyslogd(8)** man pages.
- The [Configuring system logging without journald or with minimized journald usage](#) Knowledgebase article.
- The [Negative effects of the RHEL default logging setup on performance and their mitigations](#) article.

CHAPTER 35. USING THE LOGGING SYSTEM ROLE

As a system administrator, you can use the Logging System Role to configure a RHEL host as a logging server to collect logs from many client systems.

35.1. THE LOGGING SYSTEM ROLE

With the Logging System Role, you can deploy logging configurations on local and remote hosts.

To apply a Logging System Role on one or more systems, you define the logging configuration in a *playbook*. A playbook is a list of one or more plays. Playbooks are human-readable, and they are written in the YAML format. For more information about playbooks, see [Working with playbooks](#) in Ansible documentation.

The set of systems that you want to configure according to the playbook is defined in an *inventory file*. For more information on creating and using inventories, see [How to build your inventory](#) in Ansible documentation.

Logging solutions provide multiple ways of reading logs and multiple logging outputs.

For example, a logging system can receive the following inputs:

- local files,
- **systemd/journal**,
- another logging system over the network.

In addition, a logging system can have the following outputs:

- logs stored in the local files in the **/var/log** directory,
- logs sent to Elasticsearch,
- logs forwarded to another logging system.

With the logging system role, you can combine the inputs and outputs to fit your scenario. For example, you can configure a logging solution that stores inputs from **journal** in a local file, whereas inputs read from files are both forwarded to another logging system and stored in the local log files.

35.2. LOGGING SYSTEM ROLE PARAMETERS

In a Logging System Role playbook, you define the inputs in the **logging_inputs** parameter, outputs in the **logging_outputs** parameter, and the relationships between the inputs and outputs in the **logging_flows** parameter. The Logging System Role processes these variables with additional options to configure the logging system. You can also enable encryption.



NOTE

Currently, the only available logging system in the Logging System Role is **Rsyslog**.

- **logging_inputs**: List of inputs for the logging solution.
 - **name**: Unique name of the input. Used in the **logging_flows**: inputs list and a part of the generated **config** file name.

- **type**: Type of the input element. The type specifies a task type which corresponds to a directory name in **roles/rsyslog/{tasks,vars}/inputs/**.
 - **basics**: Inputs configuring inputs from **systemd** journal or **unix** socket.
 - **kernel_message**: Load **imklog** if set to **true**. Default to **false**.
 - **use_imuxsock**: Use **imuxsock** instead of **imjournal**. Default to **false**.
 - **ratelimit_burst**: Maximum number of messages that can be emitted within **ratelimit_interval**. Default to **20000** if **use_imuxsock** is false. Default to **200** if **use_imuxsock** is true.
 - **ratelimit_interval**: Interval to evaluate **ratelimit_burst**. Default to 600 seconds if **use_imuxsock** is false. Default to 0 if **use_imuxsock** is true. 0 indicates rate limiting is turned off.
 - **persist_state_interval**: Journal state is persisted every **value** messages. Default to **10**. Effective only when **use_imuxsock** is false.
 - **files**: Inputs configuring inputs from local files.
 - **remote**: Inputs configuring inputs from the other logging system over network.
- **state**: State of the configuration file. **present** or **absent**. Default to **present**.
- **logging_outputs**: List of outputs for the logging solution.
 - **files**: Outputs configuring outputs to local files.
 - **forwards**: Outputs configuring outputs to another logging system.
 - **remote_files**: Outputs configuring outputs from another logging system to local files.
- **logging_flows**: List of flows that define relationships between **logging_inputs** and **logging_outputs**. The **logging_flows** variable has the following keys:
 - **name**: Unique name of the flow
 - **inputs**: List of **logging_inputs** name values
 - **outputs**: List of **logging_outputs** name values.

Additional resources

- Documentation installed with the **rhel-system-roles** package in **/usr/share/ansible/roles/rhel-system-roles.logging/README.html**

35.3. APPLYING A LOCAL LOGGING SYSTEM ROLE

Follow these steps to prepare and apply a Red Hat Ansible Engine playbook to configure a logging solution on a set of separate machines. Each machine will record logs locally.

Prerequisites

- Access and permissions to one or more *managed nodes*, which are systems you want to configure with the Logging System Role.

- Access and permissions to a *control node*, which is a system from which Red Hat Ansible Engine configures other systems.

On the control node:

- Red Hat Ansible Engine is installed.
- The **rhel-system-roles** package is installed.
- An inventory file which lists the managed nodes.



NOTE

You do not have to have the **rsyslog** package installed, because the system role installs **rsyslog** when deployed.

Procedure

1. Create a playbook that defines the required role:
 - a. Create a new YAML file and open it in a text editor, for example:

```
# vi logging-playbook.yml
```

- b. Insert the following content:

```
---
- name: Deploying basics input and implicit files output
  hosts: all
  roles:
    - linux-system-roles.logging
  vars:
    logging_inputs:
      - name: system_input
        type: basics
    logging_outputs:
      - name: files_output
        type: files
    logging_flows:
      - name: flow1
        inputs: [system_input]
        outputs: [files_output]
```

2. Run the playbook on a specific inventory:

```
# ansible-playbook -i inventory-file /path/to/file/logging-playbook.yml
```

Where: * **inventory-file** is the inventory file. * **logging-playbook.yml** is the playbook you use.

Verification

1. Test the syntax of the **/etc/rsyslog.conf** file:

```
# rsyslogd -N 1
rsyslogd: version 8.1911.0-6.el8, config validation run (level 1), master config
```

```
/etc/rsyslog.conf
rsyslogd: End of config validation run. Bye.
```

2. Verify that the system sends messages to the log:

- a. Send a test message:

```
# logger test
```

- b. View the **/var/log/messages** log, for example:

```
# cat /var/log/messages
Aug  5 13:48:31 hostname root[6778]: test
```

Where `hostname` is the host name of the client system. Note that the log contains the user name of the user that entered the logger command, in this case **root**.

35.4. FILTERING LOGS IN A LOCAL LOGGING SYSTEM ROLE

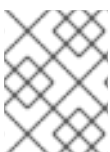
You can deploy a logging solution which filters the logs based on the **rsyslog** property-based filter.

Prerequisites

- Access and permissions to one or more *managed nodes*, which are systems you want to configure with the Logging System Role.
- Access and permissions to a *control node*, which is a system from which Red Hat Ansible Engine configures other systems.

On the control node:

- Red Hat Ansible Engine is installed
- The **rhel-system-roles** package is installed
- An inventory file which lists the managed nodes.



NOTE

You do not have to have the **rsyslog** package installed, because the system role installs **rsyslog** when deployed.

Procedure

1. Create a new **playbook.yml** file with the following content:

```
---
- name: Deploying files input and configured files output
  hosts: all
  roles:
    - linux-system-roles.logging
  vars:
    logging_inputs:
      - name: files_input0
        type: files
```

```

    input_log_path: /var/log/containerA/*.log
  - name: files_input1
    type: files
    input_log_path: /var/log/containerB/*.log
logging_outputs:
  - name: files_output0
    type: files
    property: msg
    property_op: contains
    property_value: error
    path: /var/log/errors.log
  - name: files_output1
    type: files
    property: msg
    property_op: "!contains"
    property_value: error
    path: /var/log/others.log
logging_flows:
  - name: flow0
    inputs: [files_input0, files_input1]
    outputs: [files_output0, files_output1]

```

Using this configuration, all messages that contain the **error** string are logged in **/var/log/errors.log**, and all other messages are logged in **/var/log/others.log**.

You can replace the **error** property value with the string by which you want to filter.

You can modify the variables according to your preferences.

- Optional: Verify playbook syntax.

```
# ansible-playbook --syntax-check playbook.yml
```

- Run the playbook on your inventory file:

```
# ansible-playbook -i inventory_file /path/to/file/playbook.yml
```

Verification

- Test the syntax of the **/etc/rsyslog.conf** file:

```

# rsyslogd -N 1
rsyslogd: version 8.1911.0-6.el8, config validation run (level 1), master config
/etc/rsyslog.conf
rsyslogd: End of config validation run. Bye.

```

- Verify that the system sends messages that contain the **error** string to the log:

- Send a test message:

```
# logger error
```

- View the **/var/log/errors.log** log, for example:

```
# cat /var/log/errors.log
Aug  5 13:48:31 hostname root[6778]: error
```

Where **hostname** is the host name of the client system. Note that the log contains the user name of the user that entered the logger command, in this case **root**.

Additional resources

- Documentation installed with the **rhel-system-roles** package in `/usr/share/ansible/roles/rhel-system-roles.logging/README.html`

35.5. APPLYING A REMOTE LOGGING SOLUTION USING THE LOGGING SYSTEM ROLE

Follow these steps to prepare and apply a Red Hat Ansible Engine playbook to configure a remote logging solution. In this playbook, one or more clients take logs from **systemd-journal** and forward them to a remote server. The server receives remote input from **remote_rsyslog** and **remote_files** and outputs the logs to local files in directories named by remote host names.

Prerequisites

- Access and permissions to one or more *managed nodes*, which are systems you want to configure with the Logging System Role.
- Access and permissions to a *control node*, which is a system from which Red Hat Ansible Engine configures other systems.

On the control node:

- Red Hat Ansible Engine is installed.
- The **rhel-system-roles** package is installed.
- An inventory file which lists the managed nodes.



NOTE

You do not have to have the **rsyslog** package installed, because the system role installs **rsyslog** when deployed.

Procedure

1. Create a playbook that defines the required role:
 - a. Create a new YAML file and open it in a text editor, for example:

```
# vi logging-playbook.yml
```

- b. Insert the following content into the file:

```
---
- name: Deploying remote input and remote_files output
  hosts: server
  roles:
    - linux-system-roles.logging
```

```

vars:
  logging_inputs:
    - name: remote_udp_input
      type: remote
      udp_ports: [ 601 ]
    - name: remote_tcp_input
      type: remote
      tcp_ports: [ 601 ]
  logging_outputs:
    - name: remote_files_output
      type: remote_files
  logging_flows:
    - name: flow_0
      inputs: [remote_udp_input, remote_tcp_input]
      outputs: [remote_files_output]

- name: Deploying basics input and forwards output
  hosts: clients
  roles:
    - linux-system-roles.logging
  vars:
    logging_inputs:
      - name: basic_input
        type: basics
    logging_outputs:
      - name: forward_output0
        type: forwards
        severity: info
        target: _host1.example.com_
        udp_port: 601
      - name: forward_output1
        type: forwards
        facility: mail
        target: _host1.example.com_
        tcp_port: 601
    logging_flows:
      - name: flows0
        inputs: [basic_input]
        outputs: [forward_output0, forward_output1]

[basic_input]
[forward_output0, forward_output1]

```

Where **host1.example.com** is the logging server.



NOTE

You can modify the parameters in the playbook to fit your needs.

**WARNING**

The logging solution works only with the ports defined in the SELinux policy of the server or client system and open in the firewall. The default SELinux policy includes ports 601, 514, 6514, 10514, and 20514. To use a different port, [modify the SELinux policy on the client and server systems](#). Configuring the firewall through system roles is not yet supported.

2. Create an inventory file that lists your servers and clients:

- a. Create a new file and open it in a text editor, for example:

```
# vi inventory.ini
```

- b. Insert the following content into the inventory file:

```
[servers]
server ansible_host=host1.example.com
[clients]
client ansible_host=host2.example.com
```

Where:

- **host1.example.com** is the logging server.
- **host2.example.com** is the logging client.

3. Run the playbook on your inventory.

```
# ansible-playbook -i /path/to/file/inventory.ini /path/to/file/_logging-playbook.yml
```

Where:

- **inventory.ini** is the inventory file.
- **logging-playbook.yml** is the playbook you created.

Verification

1. On both the client and the server system, test the syntax of the **/etc/rsyslog.conf** file:

```
# rsyslogd -N 1
rsyslogd: version 8.1911.0-6.el8, config validation run (level 1), master config
/etc/rsyslog.conf
rsyslogd: End of config validation run. Bye.
```

2. Verify that the client system sends messages to the server:

- a. On the client system, send a test message:

■

```
# logger test
```

- b. On the server system, view the **/var/log/messages** log, for example:

```
# cat /var/log/messages  
Aug  5 13:48:31 host2.example.com root[6778]: test
```

Where ***host2.example.com*** is the host name of the client system. Note that the log contains the user name of the user that entered the logger command, in this case **root**.

Additional resources

- [Getting started with RHEL System Roles](#)
- Documentation installed with the **rhel-system-roles** package in **/usr/share/ansible/roles/rhel-system-roles.logging/README.html**
- [RHEL System Roles](#) KB article

35.6. ADDITIONAL RESOURCES

- [Getting started with RHEL System Roles](#).
- Documentation installed with the **rhel-system-roles** package in **/usr/share/ansible/roles/rhel-system-roles.logging/README.html**.
- [RHEL System Roles](#).
- **ansible-playbook(1)** man page.

CHAPTER 36. USING THE LOGGING SYSTEM ROLES WITH RELP

Reliable Event Logging Protocol (RELP) is a networking protocol for data and message logging over the TCP network. It ensures reliable delivery of event messages and you can use it in environments that do not tolerate any message loss.

The RELP sender transfers log entries in form of commands and the receiver acknowledges them once they are processed. To ensure consistency, RELP stores the transaction number to each transferred command for any kind of message recovery.

You can consider a remote logging system in between the RELP Client and RELP Server. the RELP Client transfers the logs to the remote logging system and the RELP Server receives all the logs sent by the remote logging system.

Administrators can use the Logging System Role to configure the logging system to reliably send and receive log entries.

36.1. CONFIGURING CLIENT LOGGING WITH RELP

You can use the Logging System Role to configure logging in RHEL systems that are logged on a local machine and can transfer logs to the remote logging system with RELP by running an Ansible playbook.

This procedure configures RELP on all hosts in the **clients** group in the Ansible inventory. The RELP configuration uses Transport Layer Security (TLS) to encrypt the message transmission for secure transfer of logs over the network.

Prerequisites

- You have permissions to run playbooks on managed nodes on which you want to configure RELP.
- The managed nodes are listed in the inventory file on the control node.
- The **ansible** and **rhel-system-roles** packages are installed on the control node.

Procedure

1. Create a **playbook.yml** file with the following content:

```
---
- name: Deploying basic input and relp output
  hosts: clients
  roles:
    - rhel-system-roles.logging
  vars:
    logging_inputs:
      - name: basic_input
        type: basics
    logging_outputs:
      - name: relp_client
        type: relp
        target: _logging.server.com_
        port: 20514
```



```

tls: true
ca_cert: /etc/pki/tls/certs/ca.pem_
cert: /etc/pki/tls/certs/client-cert.pem_
private_key: /etc/pki/tls/private/client-key.pem_
pki_authmode: name
permitted_servers:
  - '*.server.example.com'
logging_flows:
  - name: _example_flow_
    inputs: [basic_input]
    outputs: [relp_client]

```

The playbooks uses following settings:

- **target:** This is a required parameter that specifies the host name where the remote logging system is running.
- **port:** Port number the remote logging system is listening.
- **tls:** Ensures secure transfer of logs over the network. If you do not want a secure wrapper you can set the **tls** variable to **false**. By default **tls** parameter is set to true while working with RELP and requires key/certificates and triplets {**ca_cert**, **cert**, **private_key**} and/or {**ca_cert_src**, **cert_src**, **private_key_src**}.
 - If {**ca_cert_src**, **cert_src**, **private_key_src**} triplet is set, the default locations **/etc/pki/tls/certs** and **/etc/pki/tls/private** are used as the destination on the managed node to transfer files from control node. In this case, the file names are identical to the original ones in the triplet
 - If {**ca_cert**, **cert**, **private_key**} triplet is set, files are expected to be on the default path before the logging configuration.
 - If both the triplets are set, files are transfered from local path from control node to specific path of the managed node.
- **ca_cert:** Represents the path to CA certificate. Default path is **/etc/pki/tls/certs/ca.pem** and the file name is set by the user.
- **cert:** Represents the path to cert. Default path is **/etc/pki/tls/certs/server-cert.pem** and the file name is set by the user.
- **private_key:** Represents the path to private key. Default path is **/etc/pki/tls/private/server-key.pem** and the file name is set by the user.
- **ca_cert_src:** Represents local CA cert file path which is copied to the target host. If **ca_cert** is specified, it is copied to the location.
- **cert_src:** Represents the local cert file path which is copied to the target host. If **cert** is specified, it is copied to the location.
- **private_key_src:** Represents the local key file path which is copied to the target host. If **private_key** is specified, it is copied to the location.
- **pki_authmode:** Accepts the authentication mode as **name** or **fingerprint**.
- **permitted_servers:** List of servers that will be allowed by the logging client to connect and send logs over TLS.

- **inputs:** List of logging input dictionary.
 - **outputs:** List of logging output dictionary.
2. Optional: Verify playbook syntax.

```
# ansible-playbook --syntax-check playbook.yml
```

3. Run the playbook:

```
# ansible-playbook -i inventory_file playbook.yml
```

36.2. CONFIGURING SERVER LOGGING WITH RELP

You can use the Logging System Role to configure logging in RHEL systems as a server and can receive logs from the remote logging system with RELP by running an Ansible playbook.

This procedure configures RELP on all hosts in the **server** group in the Ansible inventory. The RELP configuration uses TLS to encrypt the message transmission for secure transfer of logs over the network.

Prerequisites

- You have permissions to run playbooks on managed nodes on which you want to configure RELP.
- The managed nodes are listed in the inventory file on the control node.
- The **ansible** and **rhel-system-roles** packages are installed on the control node.

Procedure

1. Create a ***playbook.yml*** file with the following content:

```
---
- name: Deploying remote input and remote_files output
  hosts: server
  roles:
    - rhel-system-roles.logging
  vars:
    logging_inputs:
      - name: relp_server
        type: relp
        port: 20514
        tls: true
        ca_cert: _/etc/pki/tls/certs/ca.pem_
        cert: _/etc/pki/tls/certs/server-cert.pem_
        private_key: _/etc/pki/tls/private/server-key.pem_
        pki_authmode: name
        permitted_clients:
          - '*_example.client.com_'
    logging_outputs:
      - name: _remote_files_output_
        type: _remote_files_
```

```
logging_flows:
- name: _example_flow_
  inputs: _relp_server_
  outputs: _remote_files_output_
```

The playbooks uses following settings:

- **port**: Port number the remote logging system is listening.
- **tls**: Ensures secure transfer of logs over the network. If you do not want a secure wrapper you can set the **tls** variable to **false**. By default **tls** parameter is set to true while working with RELP and requires key/certificates and triplets {**ca_cert**, **cert**, **private_key**} and/or {**ca_cert_src**, **cert_src**, **private_key_src**}.
 - If {**ca_cert_src**, **cert_src**, **private_key_src**} triplet is set, the default locations **/etc/pki/tls/certs** and **/etc/pki/tls/private** are used as the destination on the managed node to transfer files from control node. In this case, the file names are identical to the original ones in the triplet
 - If {**ca_cert**, **cert**, **private_key**} triplet is set, files are expected to be on the default path before the logging configuration.
 - If both the triplets are set, files are transferred from local path from control node to specific path of the managed node.
- **ca_cert**: Represents the path to CA certificate. Default path is **/etc/pki/tls/certs/ca.pem** and the file name is set by the user.
- **cert**: Represents the path to cert. Default path is **/etc/pki/tls/certs/server-cert.pem** and the file name is set by the user.
- **private_key**: Represents the path to private key. Default path is **/etc/pki/tls/private/server-key.pem** and the file name is set by the user.
- **ca_cert_src**: Represents local CA cert file path which is copied to the target host. If **ca_cert** is specified, it is copied to the location.
- **cert_src**: Represents the local cert file path which is copied to the target host. If **cert** is specified, it is copied to the location.
- **private_key_src**: Represents the local key file path which is copied to the target host. If **private_key** is specified, it is copied to the location.
- **pki_authmode**: Accepts the authentication mode as **name** or **fingerprint**.
- **permitted_clients**: List of clients that will be allowed by the logging server to connect and send logs over TLS.
- **inputs**: List of logging input dictionary.
- **outputs**: List of logging output dictionary.

2. Optional: Verify playbook syntax.

```
# ansible-playbook --syntax-check playbook.yml
```

3. Run the playbook:

```
# ansible-playbook -i inventory_file playbook.yml
```

CHAPTER 37. USING THE LOGGING SYSTEM ROLE WITH TLS

Transport Layer Security (TLS) is a cryptographic protocol designed to securely communicate over the computer network.

As an administrator, you can use the Logging System Role on RHEL to configure secure transfer of logs using Red Hat Ansible Automation Platform.

37.1. CONFIGURING CLIENT LOGGING WITH TLS

You can use the Logging System Role to configure logging in RHEL systems that are logged on a local machine and can transfer logs to the remote logging system with TLS by running an Ansible playbook.

This procedure configures TLS on all hosts in the clients group in the Ansible inventory. The TLS protocol encrypts the message transmission for secure transfer of logs over the network.

Prerequisites

- You have permissions to run playbooks on managed nodes on which you want to configure TLS.
- The managed nodes are listed in the inventory file on the control node.
- The **ansible** and **rhel-system-roles** packages are installed on the control node.

Procedure

1. Create a **playbook.yml** file with the following content:

```
---
- name: Deploying files input and forwards output with certs
  hosts: clients
  roles:
    - rhel-system-roles.logging
  vars:
    logging_pki_files:
      - ca_cert_src: /local/path/to/ca_cert.pem
        cert_src: /local/path/to/cert.pem
        private_key_src: /local/path/to/key.pem
    logging_inputs:
      - name: input_name
        type: files
        input_log_path: /var/log/containers/*.log
    logging_outputs:
      - name: output_name
        type: forwards
        target: your_target_host
        tcp_port: 514
        tls: true
        pki_authmode: x509/name
        permitted_server: 'server.example.com'
    logging_flows:
      - name: flow_name
        inputs: [input_name]
        outputs: [output_name]
```

The playbook uses the following parameters:

logging_pki_files

Using this parameter you can configure TLS and has to pass **ca_cert_src**, **cert_src**, and **private_key_src** parameters.

ca_cert

Represents the path to CA certificate. Default path is **/etc/pki/tls/certs/ca.pem** and the file name is set by the user.

cert

Represents the path to cert. Default path is **/etc/pki/tls/certs/server-cert.pem** and the file name is set by the user.

private_key

Represents the path to private key. Default path is **/etc/pki/tls/private/server-key.pem** and the file name is set by the user.

ca_cert_src

Represents local CA cert file path which is copied to the target host. If **ca_cert** is specified, it is copied to the location.

cert_src

Represents the local cert file path which is copied to the target host. If **cert** is specified, it is copied to the location.

private_key_src

Represents the local key file path which is copied to the target host. If **private_key** is specified, it is copied to the location.

tls

Using this parameter ensures secure transfer of logs over the network. If you do not want a secure wrapper, you can set **tls: true**.

2. Verify playbook syntax:

```
# ansible-playbook --syntax-check playbook.yml
```

3. Run the playbook on your inventory file:

```
# ansible-playbook -i inventory_file playbook.yml
```

37.2. CONFIGURING SERVER LOGGING WITH TLS

You can use the Logging System Role to configure logging in RHEL systems as a server and can receive logs from the remote logging system with TLS by running an Ansible playbook.

This procedure configures TLS on all hosts in the server group in the Ansible inventory.

Prerequisites

- You have permissions to run playbooks on managed nodes on which you want to configure TLS.
- The managed nodes are listed in the inventory file on the control node.
- The **ansible** and **rhel-system-roles** packages are installed on the control node.

Procedure

1. Create a **playbook.yml** file with the following content:

```
---
- name: Deploying remote input and remote_files output with certs
  hosts: server
  roles:
    - rhel-system-roles.logging
  vars:
    logging_pki_files:
      - ca_cert_src: /local/path/to/ca_cert.pem
        cert_src: /local/path/to/cert.pem
        private_key_src: /local/path/to/key.pem
    logging_inputs:
      - name: input_name
        type: remote
        tcp_ports: 514
        tls: true
        permitted_clients: ['clients.example.com']
    logging_outputs:
      - name: output_name
        type: remote_files
        remote_log_path: /var/log/remote/%FROMHOST%/PROGRAMNAME:::secpath-
replace%.log
        async_writing: true
        client_count: 20
        io_buffer_size: 8192
    logging_flows:
      - name: flow_name
        inputs: [input_name]
        outputs: [output_name]
```

The playbook uses the following parameters:

logging_pki_files

Using this parameter you can configure TLS and has to pass **ca_cert_src**, **cert_src**, and **private_key_src** parameters.

ca_cert

Represents the path to CA certificate. Default path is **/etc/pki/tls/certs/ca.pem** and the file name is set by the user.

cert

Represents the path to cert. Default path is **/etc/pki/tls/certs/server-cert.pem** and the file name is set by the user.

private_key

Represents the path to private key. Default path is **/etc/pki/tls/private/server-key.pem** and the file name is set by the user.

ca_cert_src

Represents local CA cert file path which is copied to the target host. If **ca_cert** is specified, it is copied to the location.

cert_src

Represents local cert file path which is copied to the target host. If **cert** is specified, it is copied to the location.

Represents the local cert file path which is copied to the target host. If **cert** is specified, it is copied to the location.

private_key_src

Represents the local key file path which is copied to the target host. If **private_key** is specified, it is copied to the location.

tls

Using this parameter ensures secure transfer of logs over the network. If you do not want a secure wrapper, you can set **tls: true**.

2. Verify playbook syntax:

```
# ansible-playbook --syntax-check playbook.yml
```

3. Run the playbook on your inventory file:

```
# ansible-playbook -i inventory_file playbook.yml
```


CHAPTER 38. INTRODUCTION TO PYTHON

Python is a high-level programming language that supports multiple programming paradigms, such as object-oriented, imperative, functional, and procedural paradigms. 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 use these packages, you must have the **python*** packages installed.

38.1. PYTHON VERSIONS

Two incompatible versions of Python are widely used, Python 2.x and Python 3.x. RHEL 8 provides the following versions of Python.

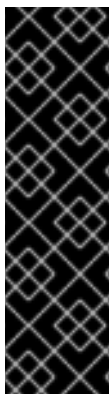
Table 38.1. Python versions in RHEL 8

Version	Package to install	Command examples	Available since	Life cycle
Python 3.6	python3	python3, pip3	RHEL 8.0	full RHEL 8
Python 2.7	python2	python2, pip2	RHEL 8.0	shorter
Python 3.8	python38	python3.8, pip3.8	RHEL 8.2	shorter
Python 3.9	python39	python3.9, pip3.9	RHEL 8.4	shorter

For details about the length of support, see [Red Hat Enterprise Linux Life Cycle](#) and [Red Hat Enterprise Linux 8 Application Streams Life Cycle](#).

Each of the Python versions is distributed in a separate module and by design you can install multiple modules in parallel on the same system.

The **python38** and **python39** modules do not include the same bindings to system tools (RPM, DNF, SELinux, and others) that are provided for the **python36** module.



IMPORTANT

Always specify the 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**, **pip2**, **pip3.8**, or **pip3.9**.

The unversioned **python** command (**/usr/bin/python**) is not available by default in RHEL 8. You can configure it using the **alternatives** command; for instructions, see [Configuring the unversioned Python](#). Any manual changes to **/usr/bin/python**, except changes made using the **alternatives** command, might be overwritten upon an update.

As a system administrator, use Python 3 for the following reasons:

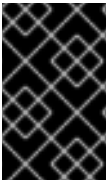
- Python 3 represents the main development direction of the Python project.

- Support for Python 2 in the upstream community ended in 2020.
- Popular Python libraries are discontinuing Python 2 support in upstream.
- Python 2 in Red Hat Enterprise Linux 8 will have a shorter life cycle and aims to facilitate a smoother transition to **Python 3** for customers.

For developers, Python 3 has the following advantages over Python 2:

- Python 3 enables you to write 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, and chained exceptions.

However, legacy software might require `/usr/bin/python` to be configured to Python 2. For this reason, no default **python** package is distributed with Red Hat Enterprise Linux 8, and you can choose between using Python 2 and 3 as `/usr/bin/python`, as described in [Configuring the unversioned Python](#).



IMPORTANT

System tools in Red Hat Enterprise Linux 8 use Python version 3.6 provided by the internal **platform-python** package. Red Hat advises customers to use the **python36** package instead.

CHAPTER 39. INSTALLING AND USING PYTHON

In Red Hat Enterprise Linux 8, Python 3 is distributed in versions 3.6, 3.8, and 3.9, provided by the **python36**, **python38**, and **python39** modules in the AppStream repository.



WARNING

Using the unversioned **python** command to install or run Python does not work by default due to ambiguity. Always specify the version of Python, or configure the system default version by using the **alternatives** command.

39.1. INSTALLING PYTHON 3

By design, you can install RHEL 8 modules in parallel, including the **python27**, **python36**, **python38**, and **python39** modules. Note that parallel installation is not supported for multiple streams within a single module.

You can install Python 3.8 and Python 3.9, including packages built for either version, in parallel with Python 3.6 on the same system, with the exception of the **mod_wsgi** module. Due to a limitation of the Apache HTTP Server, only one of the **python3-mod_wsgi**, **python38-mod_wsgi**, or **python39-mod_wsgi** packages can be installed on a system.

Procedure

- To install Python 3.6 from the **python36** module, use:

```
# yum install python3
```

The **python36:3.6** module stream is enabled automatically.

- To install Python 3.8 from the **python38** module, use:

```
# yum install python38
```

The **python38:3.8** module stream is enabled automatically.

- To install Python 3.9 from the **python39** module, use:

```
# yum install python39
```

The **python39:3.9** module stream is enabled automatically.

Verification steps

- To verify the Python version installed on your system, use the **--version** option with the **python** command specific for your required version of Python.
 - For Python 3.6:

```
$ python3 --version
```

- For Python 3.8:

```
$ python3.8 --version
```

- For Python 3.9:

```
$ python3.9 --version
```

Additional resources

- [Installing, managing, and removing user-space components](#)

39.2. INSTALLING ADDITIONAL PYTHON 3 PACKAGES

Packages with add-on modules for Python 3.6 generally use the **python3-** prefix, packages for Python 3.8 include the **python38-** prefix, and packages for Python 3.9 include the **python39-** prefix. Always include the prefix when installing additional Python packages, as shown in the examples below.

Procedure

- To install the **Requests** module for Python 3.6, use:

```
# yum install python3-requests
```

- To install the **Cython** extension to Python 3.8, use:

```
# yum install python38-Cython
```

- To install the **pip** package installer from Python 3.9, use:

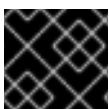
```
# yum install python39-pip
```

39.3. INSTALLING ADDITIONAL PYTHON 3 TOOLS FOR DEVELOPERS

Additional Python tools for developers are distributed through the CodeReady Linux Builder repository in the respective **python3x-devel** module.

The **python38-devel** module contains the **python38-pytest** package and its dependencies: the **pyparsing**, **atomicwrites**, **attrs**, **packaging**, **py**, **more-itertools**, **pluggy**, and **wcwidth** packages.

The **python39-devel** module contains the **python39-pytest** package and its dependencies: the **pyparsing**, **attrs**, **packaging**, **py**, **more-itertools**, **pluggy**, **wcwidth**, **iniconfig**, and **pybind11** packages. The **python39-devel** module also contains the **python39-debug** and **python39-Cython** packages.



IMPORTANT

The CodeReady Linux Builder repository and its content is unsupported by Red Hat.

To install packages from the **python39-devel** module, use the following the procedure.

Procedure

1. Enable the CodeReady Linux Builder repository:

```
# subscription-manager repos --enable codeready-builder-for-rhel-8-x86_64-rpms
```

2. Enable the **python39-devel** module:

```
# yum module enable python39-devel
```

3. Install the **python39-pytest** package:

```
# yum install python39-pytest
```

To install packages from the **python38-devel** module, replace *python39-* with *python38-* in the commands above.

Additional resources

- [How to enable and make use of content within CodeReady Linux Builder](#)

39.4. INSTALLING PYTHON 2

Some applications and scripts have not yet been fully ported to Python 3 and require Python 2 to run. 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.



WARNING

Note that Python 3 is the main development direction of the Python project. Support for Python 2 is being phased out. The **python27** module has a shorter support period than Red Hat Enterprise Linux 8.

Procedure

- To install Python 2.7 from the **python27** module, use:

```
# yum install python2
```

The **python27:2.7** module stream is enabled automatically.

Packages with add-on modules for Python 2 generally use the **python2-** prefix. Always include the prefix when installing additional Python packages, as shown in the examples below.

- To install the **Requests** module for Python 2, use:

```
# yum install python2-requests
```

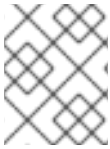
- To install the **Cython** extension to Python 2, use:

```
# yum install python2-Cython
```

Verification steps

- To verify the Python version installed on your system, use:

```
$ python2 --version
```



NOTE

By design, you can install RHEL 8 modules in parallel, including the **python27**, **python36**, **python38**, and **python39** modules.

Additional resources

- [Installing, managing, and removing user-space components in RHEL 8](#)

39.5. 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.

39.6. USING PYTHON

When running the Python interpreter or Python-related commands, always specify the version.

Prerequisites

- Ensure that the required version of Python is installed.

Procedure

- To run the Python 3.6 interpreter or related commands, use, for example:

```
$ python3
$ python3 -m cython --help
$ pip3 install package
```

- To run the Python 3.8 interpreter or related commands, use, for example:

```
$ python3.8
$ python3.8 -m cython --help
$ pip3.8 install package
```

- To run the Python 3.9 interpreter or related commands, use, for example:

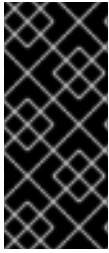
```
$ python3.9  
$ python3.9 -m pip --help  
$ pip3.9 install package
```

- To run the Python 2 interpreter or related commands, use, for example:

```
$ python2  
$ python2 -m cython --help  
$ pip2 install package
```

CHAPTER 40. CONFIGURING THE UNVERSIONED PYTHON

System administrators can configure the unversioned **python** command, located at `/usr/bin/python`, using the **alternatives** command. Note that the required package, **python3**, **python38**, **python39**, or **python2**, must be installed before configuring the unversioned command to the respective version.



IMPORTANT

The `/usr/bin/python` executable is controlled by the **alternatives** system. Any manual changes may be overwritten upon an update.

Additional Python-related commands, such as **pip3**, do not have configurable unversioned variants.

40.1. CONFIGURING THE UNVERSIONED PYTHON COMMAND DIRECTLY

You can configure the unversioned **python** command directly to a selected version of Python.

Prerequisites

- Ensure that the required version of Python is installed.

Procedure

- To configure the unversioned **python** command to Python 3.6, use:

```
# alternatives --set python /usr/bin/python3
```

- To configure the unversioned **python** command to Python 3.8, use:

```
# alternatives --set python /usr/bin/python3.8
```

- To configure the unversioned **python** command to Python 3.9, use:

```
# alternatives --set python /usr/bin/python3.9
```

- To configure the unversioned **python** command to Python 2, use:

```
# alternatives --set python /usr/bin/python2
```

40.2. CONFIGURING THE UNVERSIONED PYTHON COMMAND TO THE REQUIRED PYTHON VERSION INTERACTIVELY

You can configure the unversioned **python** command to the required Python version interactively.

Prerequisites

- Ensure that the required version of Python is installed.

Procedure

1. To configure the unversioned **python** command interactively, use:

```
# alternatives --config python
```

2. Select the required version from the provided list.
3. To reset this configuration and remove the unversioned **python** command, use:

```
# alternatives --auto python
```

40.3. ADDITIONAL RESOURCES

- **alternatives(8)** and **unversioned-python(1)** man pages

CHAPTER 41. PACKAGING PYTHON 3 RPMS

Most Python projects use Setuptools for packaging, and define package information in the **setup.py** file. For more information about Setuptools packaging, see the [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.

41.1. SPEC FILE DESCRIPTION FOR A PYTHON 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)

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 prefix determining the version, for example, **python3** for Python 3.6, **python38** for Python 3.8, or **python39** for Python 3.9.

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.

```
%global modname detox 1

Name:      python3-detox 2
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 3
BuildRequires: python3-setuptools
BuildRequires: python36-rpm-macros
BuildRequires: python3-six
BuildRequires: python3-tox
BuildRequires: python3-py
BuildRequires: python3-eventlet

%?python_enable_dependency_generator 4

%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 CHANGELOG
%license LICENSE
%{_bindir}/detox
%{python3_sitelib}/%{modname}/
%{python3_sitelib}/%{modname}-%{version}*

%changelog
...
```

5

6

- 1 The **modname** macro contains the name of the Python project. In this example it is **detox**.
- 2 When packaging a Python project into RPM, the **python3** prefix always needs to be added to the original name of the project. The original name here is **detox** and the **name of the RPM** is **python3-detox**.
- 3 **BuildRequires** specifies what packages are required to build and test this package. In **BuildRequires**, always include items providing tools necessary for building Python packages: **python36-devel** and **python3-setuptools**. The **python36-rpm-macros** package is required so that files with **/usr/bin/python3** interpreter directives are automatically changed to **/usr/bin/python3.6**.
- 4 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 **%python_enable_dependency_generator** macro to automatically use dependencies defined in the **setup.py** file. If a package has dependencies that are not specified using **Setuptools**, specify them within additional **Requires** directives.
- 5 The **%py3_build** and **%py3_install** macros run the **setup.py build** and **setup.py install** commands, respectively, with additional arguments to specify installation locations, the interpreter to use, and other details.
- 6 The **check** section provides a macro that runs the correct version of Python. The **%{__python3}** macro contains a path for the Python 3 interpreter, for example **/usr/bin/python3**. We recommend to always use the macro rather than a literal path.

41.2. COMMON MACROS FOR PYTHON 3 RPMS

In a SPEC file, always use the macros that are described in the following Macros for Python 3 RPMS table rather than hardcoding their values.

In macro names, always use **python3** or **python2** instead of unversioned **python**. Configure the particular Python 3 version in the **BuildRequires** of the SPEC file to **python36-rpm-macros**, **python38-rpm-macros**, or **python39-rpm-macros**.

Table 41.1. Macros for Python 3 RPMS

Macro	Normal Definition	Description
<code>%{__python3}</code>	<code>/usr/bin/python3</code>	Python 3 interpreter
<code>%{python3_version}</code>	3.6	The full version of the Python 3 interpreter.
<code>%{python3_sitelib}</code>	<code>/usr/lib/python3.6/site-packages</code>	Where pure-Python modules are installed.
<code>%{python3_sitelib64}</code>	<code>/usr/lib64/python3.6/site-packages</code>	Where modules containing architecture-specific extensions are installed.
<code>%py3_build</code>		Runs the setup.py build command with arguments suitable for a system package.
<code>%py3_install</code>		Runs the setup.py install command with arguments suitable for a system package.

41.3. AUTOMATIC PROVIDES FOR PYTHON RPMS

When packaging a Python project, make sure that the following directories are included in the resulting RPM if these directories are present:

- **.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.

CHAPTER 42. HANDLING INTERPRETER DIRECTIVES IN PYTHON SCRIPTS

In Red Hat Enterprise Linux 8, executable Python scripts are expected to use interpreter directives (also known as hashbangs or shebangs) that explicitly specify at a minimum the major Python version. For example:

```
#!/usr/bin/python3
#!/usr/bin/python3.6
#!/usr/bin/python2
```

The **/usr/lib/rpm/redhat/brp-mangle-shebangs** buildroot policy (BRP) script is run automatically when building any RPM package, and attempts to correct interpreter directives in all executable files.

The BRP script generates errors when encountering a Python script with an ambiguous interpreter directive, such as:

```
#!/usr/bin/python
```

or

```
#!/usr/bin/env python
```

42.1. MODIFYING INTERPRETER DIRECTIVES IN PYTHON SCRIPTS

Modify interpreter directives in the Python scripts that cause the build errors at RPM build time.

Prerequisites

- Some of the interpretive directives in your Python scripts cause a build error.

Procedure

To modify interpreter directives, complete one of the following tasks:

- Apply the **pathfix.py** script from the **platform-python-devel** package:

```
# pathfix.py -pn -i %(__python3) PATH ...
```

Note that 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 interpreter directive. Add this command to the **%prep** section or at the end of the **%install** section.

- 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 an RPM build, replace **%(__python3)** from the example above with a path for the interpreter directive, such as **/usr/bin/python3**.

If the packaged Python scripts require a version other than Python 3.6, adjust the preceding commands to include the required version.

42.2. CHANGING /usr/bin/python3 INTERPRETER DIRECTIVES IN YOUR CUSTOM PACKAGES

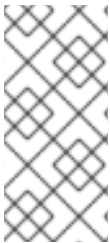
By default, interpreter directives in the form of **/usr/bin/python3** are replaced with interpreter directives pointing to Python from the **platform-python** package, which is used for system tools with Red Hat Enterprise Linux. You can change the **/usr/bin/python3** interpreter directives in your custom packages to point to a specific version of Python that you have installed from the AppStream repository.

Procedure

- To build your package for a specific version of Python, add the **python*-rpm-macros** subpackage of the respective **python** package to the **BuildRequires** section of the SPEC file. For example, for Python 3.6, include the following line:

```
BuildRequires: python36-rpm-macros
```

As a result, the **/usr/bin/python3** interpreter directives in your custom package are automatically converted to **/usr/bin/python3.6**.



NOTE

To prevent the BRP script from checking and modifying interpreter directives, use the following RPM directive:

```
%undefine %brp_mangle_shebangs
```

CHAPTER 43. USING THE PHP SCRIPTING LANGUAGE

Hypertext Preprocessor (PHP) is a general-purpose scripting language mainly used for server-side scripting, which enables you to run the PHP code using a web server.

In RHEL 8, the PHP scripting language is provided by the **php** module, which is available in multiple streams (versions).

Depending on your use case, you can install a specific profile of the selected module stream:

- **common** - The default profile for server-side scripting using a web server. It includes several widely used extensions.
- **minimal** - This profile installs only the command-line interface for scripting with PHP without using a web server.
- **devel** - This profile includes packages from the **common** profile and additional packages for development purposes.

43.1. INSTALLING THE PHP SCRIPTING LANGUAGE

This section describes how to install a selected version of the **php** module.

Procedure

- To install a **php** module stream with the default profile, use:

```
# yum module install php:stream
```

Replace *stream* with the version of PHP you wish to install.

For example, to install PHP 7.4:

```
# yum module install php:7.4
```

The default **common** profile installs also the **php-fpm** package, and preconfigures PHP for use with the **Apache HTTP Server** or **nginx**.

- To install a specific profile of a **php** module stream, use:

```
# yum module install php:stream/profile
```

Replace *stream* with the desired version and *profile* with the name of the profile you wish to install.

For example, to install PHP 7.4 for use without a web server:

```
# yum module install php:7.4/minimal
```

Additional resources

- If you want to upgrade from an earlier version of PHP available in RHEL 8, see [Switching to a later stream](#).

- For more information on managing RHEL 8 modules and streams, see [Installing, managing, and removing user-space components](#).

43.2. USING THE PHP SCRIPTING LANGUAGE WITH A WEB SERVER

43.2.1. Using PHP with the Apache HTTP Server

In Red Hat Enterprise Linux 8, the **Apache HTTP Server** enables you to run PHP as a FastCGI process server. FastCGI Process Manager (FPM) is an alternative PHP FastCGI daemon that allows a website to manage high loads. PHP uses FastCGI Process Manager by default in RHEL 8.

This section describes how to run the PHP code using the FastCGI process server.

Prerequisites

- The PHP scripting language is installed on your system.
See
- [Installing the PHP scripting language](#)

Procedure

1. Install the **httpd** module:

```
# yum module install httpd:2.4
```

2. Start the **Apache HTTP Server**:

```
# systemctl start httpd
```

Or, if the **Apache HTTP Server** is already running on your system, restart the **httpd** service after installing PHP:

```
# systemctl restart httpd
```

3. Start the **php-fpm** service:

```
# systemctl start php-fpm
```

4. Optional: Enable both services to start at boot time:

```
# systemctl enable php-fpm httpd
```

5. To obtain information about your PHP settings, create the **index.php** file with the following content in the **/var/www/html/** directory:

```
echo '<?php phpinfo(); ?>' > /var/www/html/index.php
```

6. To run the **index.php** file, point the browser to:

```
http://<hostname>/
```


7. Optional: Adjust configuration if you have specific requirements:

- `/etc/httpd/conf/httpd.conf` – generic **httpd** configuration
- `/etc/httpd/conf.d/php.conf` – PHP-specific configuration for **httpd**
- `/usr/lib/systemd/system/httpd.service.d/php-fpm.conf` – by default, the **php-fpm** service is started with **httpd**
- `/etc/php-fpm.conf` – FPM main configuration
- `/etc/php-fpm.d/www.conf` – default **www** pool configuration

Example 43.1. Running a "Hello, World!" PHP script using the Apache HTTP Server

1. Create a **hello** directory for your project in the `/var/www/html/` directory:

```
# mkdir hello
```

2. Create a **hello.php** file in the `/var/www/html/hello/` directory with the following content:

```
# <!DOCTYPE html>
<html>
<head>
<title>Hello, World! Page</title>
</head>
<body>
<?php
    echo 'Hello, World!';
?>
</body>
</html>
```

3. Start the **Apache HTTP Server**:

```
# systemctl start httpd
```

4. To run the **hello.php** file, point the browser to:

```
http://<hostname>/hello/hello.php
```

As a result, a web page with the "Hello, World!" text is displayed.

Additional resources

- [Setting up the Apache HTTP web server](#)

43.2.2. Using PHP with the nginx web server

This section describes how to run PHP code through the **nginx** web server.

Prerequisites

- The PHP scripting language is installed on your system.
See
- [Installing the PHP scripting language](#)

Procedure

1. Install an **nginx** module stream:

```
# yum module install nginx:stream
```

Replace *stream* with the version of **nginx** you wish to install.

For example, to install **nginx** version 1.18:

```
# yum module install nginx:1.18
```

2. Start the **nginx** server:

```
# systemctl start nginx
```

Or, if the **nginx** server is already running on your system, restart the **nginx** service after installing PHP:

```
# systemctl restart nginx
```

3. Start the **php-fpm** service:

```
# systemctl start php-fpm
```

4. Optional: Enable both services to start at boot time:

```
# systemctl enable php-fpm nginx
```

5. To obtain information about your PHP settings, create the **index.php** file with the following content in the **/usr/share/nginx/html/** directory:

```
echo '<?php phpinfo(); ?>' > /usr/share/nginx/html/index.php
```

6. To run the **index.php** file, point the browser to:

```
http://<hostname>/
```

7. Optional: Adjust configuration if you have specific requirements:

- **/etc/nginx/nginx.conf** - **nginx** main configuration
- **/etc/nginx/conf.d/php-fpm.conf** - FPM configuration for **nginx**
- **/etc/php-fpm.conf** - FPM main configuration
- **/etc/php-fpm.d/www.conf** - default **www** pool configuration

Example 43.2. Running a "Hello, World!" PHP script using the nginx server

1. Create a **hello** directory for your project in the `/usr/share/nginx/html/` directory:

```
# mkdir hello
```

2. Create a **hello.php** file in the `/usr/share/nginx/html/hello/` directory with the following content:

```
# <!DOCTYPE html>
<html>
<head>
<title>Hello, World! Page</title>
</head>
<body>
<?php
    echo 'Hello, World!';
?>
</body>
</html>
```

3. Start the **nginx** server:

```
# systemctl start nginx
```

4. To run the **hello.php** file, point the browser to:

```
http://<hostname>/hello/hello.php
```

As a result, a web page with the "Hello, World!" text is displayed.

Additional resources

- [Setting up and configuring NGINX](#)

43.3. RUNNING A PHP SCRIPT USING THE COMMAND-LINE INTERFACE

A PHP script is usually run using a web server, but also can be run using the command-line interface.

If you want to run **php** scripts using only command-line, install the **minimal** profile of a **php** module stream.

See

- [Installing the PHP scripting language](#)

Prerequisites

- The PHP scripting language is installed on your system.
See

- [Installing the PHP scripting language](#)

Procedure

1. In a text editor, create a ***filename.php*** file
Replace *filename* with the name of your file.
2. Execute the created ***filename.php*** file from the command line:

```
# php filename.php
```

Example 43.3. Running a "Hello, World!" PHP script using the command-line interface

1. Create a **hello.php** file with the following content using a text editor:

```
<?php
    echo 'Hello, World!';
?>
```

2. Execute the **hello.php** file from the command line:

```
# php hello.php
```

As a result, "Hello, World!" is printed.

43.4. ADDITIONAL RESOURCES

- **httpd(8)** – The manual page for the **httpd** service containing the complete list of its command-line options.
- **httpd.conf(5)** – The manual page for **httpd** configuration, describing the structure and location of the **httpd** configuration files.
- **nginx(8)** – The manual page for the **nginx** web server containing the complete list of its command-line options and list of signals.
- **php-fpm(8)** – The manual page for PHP FPM describing the complete list of its command-line options and configuration files.

CHAPTER 44. USING 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. If these prerequisites are fulfilled, the language meta-packages pull their langpack for the selected language automatically in the transaction set.

Prerequisites

- 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.

For more information, see

- [Checking languages that provide langpacks](#)
- The base package, for which you want to search the locale packages, has already been installed on the system.

44.1. CHECKING LANGUAGES THAT PROVIDE LANGPACKS

Follow this procedure to check which languages provide langpacks.

Procedure

- Execute the following command:

```
# yum list langpacks-*
```

44.2. WORKING WITH RPM WEAK DEPENDENCY-BASED LANGPACKS

This section describes multiple actions that you may want to perform when querying RPM weak dependency-based langpacks, installing or removing language support.

44.2.1. Listing already installed language support

To list the already installed language support, use this procedure.

Procedure

- Execute the following command:

```
# yum list installed langpacks*
```

44.2.2. Checking the availability of language support

To check if language support is available for any language, use the following procedure.

Procedure

- Execute the following command:

```
# yum list available langpacks*
```

44.2.3. Listing packages installed for a language

To list what packages get installed for any language, use the following procedure:

Procedure

- Execute the following command:

```
# yum repoquery --whatsupplements langpacks-<locale_code>
```

44.2.4. Installing language support

To add new a language support, use the following procedure.

Procedure

- Execute the following command:

```
# yum install langpacks-<locale_code>
```

44.2.5. Removing language support

To remove any installed language support, use the following procedure.

Procedure

- Execute the following command:

```
# yum remove langpacks-<locale_code>
```

44.3. 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>**).

To install locales individually, and thus gain a smaller package installation footprint, use the following procedure.

Procedure

- Execute the following command:

```
# 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 45. GETTING STARTED WITH TCL/TK

45.1. 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](#).

45.2. NOTABLE CHANGES IN 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.

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](#).

45.3. 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
- Users scripting tasks with **Tcl/Tk**

45.3.1. Migration path for developers of Tcl extensions

To make your code compatible with **Tcl 8.6**, use the following procedure.

Procedure

1. Rewrite the code to use the **interp** structure. For example, if your code reads **interp** → **errorLine**, rewrite it to use the following function:

```
Tcl_GetErrorLine(interp)
```

This is necessary because **Tcl 8.6** limits direct access to members of the **interp** structure.

2. 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:

```
# include <tcl.h>
# if !defined(Tcl_GetErrorLine)
# define Tcl_GetErrorLine(interp) (interp->errorLine)
# endif
```

45.3.2. 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**.

To migrate you code into **Tcl 8.6**, use this procedure.

Procedure

- When writing a portable code, make sure to not use the commands that are no longer supported in **Tk 8.6**:

```
tklconList_Arrange
tklconList_AutoScan
tklconList_Btn1
tklconList_Config
tklconList_Create
tklconList_CtrlBtn1
tklconList_Curselection
tklconList_DeleteAll
tklconList_Double1
tklconList_DrawSelection
tklconList_FocusIn
tklconList_FocusOut
tklconList_Get
tklconList_Goto
tklconList_Index
tklconList_Invoke
tklconList_KeyPress
tklconList_Leave1
tklconList_LeftRight
tklconList_Motion1
tklconList_Reset
tklconList_ReturnKey
tklconList_See
tklconList_Select
tklconList_Selection
tklconList_ShiftBtn1
tklconList_UpDown
```

Note that you can check the list of unsupported commands also in the **/usr/share/tk8.6/unsupported.tcl** file.