Red Hat Enterprise Linux 9

Configuring basic system settings

A guide to configuring basic system settings in Red Hat Enterprise Linux 9
Abstract

This document describes basics of system administration on Red Hat Enterprise Linux 9. 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 DNF, using systemd for service management, managing users, groups and file permissions, using chrony to configure NTP and others.
# Table of Contents

**MAKING OPEN SOURCE MORE INCLUSIVE** ................................................................. 7

**PROVIDING FEEDBACK ON RED HAT DOCUMENTATION** ........................................ 8

**CHAPTER 1. GETTING STARTED WITH RHEL SYSTEM ROLES** ..................................... 9
   1.1. INTRODUCTION TO RHEL SYSTEM ROLES ....................................................... 9
   1.2. RHEL SYSTEM ROLES TERMINOLOGY ............................................................ 9
   1.3. APPLYING A ROLE ....................................................................................... 10
   1.4. ADDITIONAL RESOURCES ........................................................................... 12

**CHAPTER 2. CHANGING BASIC ENVIRONMENT SETTINGS** ......................................... 13
   2.1. CONFIGURING THE DATE AND TIME .............................................................. 13
      2.1.1. Displaying the current date and time ....................................................... 13
   2.2. CONFIGURING THE SYSTEM LOCALE ............................................................. 13
   2.3. CONFIGURING THE KEYBOARD LAYOUT ....................................................... 14
   2.4. CHANGING THE LANGUAGE USING DESKTOP GUI ....................................... 15
   2.5. ADDITIONAL RESOURCES ........................................................................... 17

**CHAPTER 3. CONFIGURING AND MANAGING NETWORK ACCESS** ............................. 18
   3.1. CONFIGURING THE NETWORK AND HOST NAME IN THE GRAPHICAL INSTALLATION MODE 18
   3.2. CONFIGURING A STATIC ETHERNET CONNECTION USING NMCLI .................. 19
   3.3. ADDING A CONNECTION PROFILE USING NMTUI ....................................... 22
   3.4. MANAGING NETWORKING IN THE RHEL WEB CONSole ............................... 24
   3.5. MANAGING NETWORKING USING RHEL SYSTEM ROLES ......................... 25
   3.6. ADDITIONAL RESOURCES ........................................................................... 26

**CHAPTER 4. REGISTERING THE SYSTEM AND MANAGING SUBSCRIPTIONS** .............. 27
   4.1. REGISTERING THE SYSTEM AFTER THE INSTALLATION ............................... 27
   4.2. REGISTERING SUBSCRIPTIONS WITH CREDENTIALS IN THE WEB CONSOLE ... 28
   4.3. REGISTERING A SYSTEM USING RED HAT ACCOUNT ON GNOME ............... 29
   4.4. REGISTERING A SYSTEM USING AN ACTIVATION KEY ON GNOME .............. 30

**CHAPTER 5. MAKING SYSTEMD SERVICES START AT BOOT TIME** ............................ 31
   5.1. ENABLING OR DISABLING THE SERVICES ..................................................... 31
   5.2. MANAGING SERVICES IN THE RHEL WEB CONSOLE .................................... 31

**CHAPTER 6. CONFIGURING SYSTEM SECURITY** ..................................................... 34
   6.1. ENABLING THE FIREWALLD SERVICE ............................................................. 34
   6.2. MANAGING BASIC SELINUX SETTINGS .......................................................... 35
   6.3. ENSURING THE REQUIRED STATE OF SELINUX ............................................ 35
   6.4. ADDITIONAL RESOURCES ........................................................................... 36

**CHAPTER 7. GETTING STARTED WITH MANAGING USER ACCOUNTS** ....................... 37
   7.1. MANAGING ACCOUNTS AND GROUPS USING COMMAND LINE TOOLS .......... 37
   7.2. SYSTEM USER ACCOUNTS MANAGED IN THE WEB CONSOLE ..................... 38
   7.3. ADDING NEW ACCOUNTS USING THE WEB CONSOLE .................................. 38

**CHAPTER 8. DUMPING A CRASHED KERNEL FOR LATER ANALYSIS** ...................... 40
   8.1. WHAT IS KDUMP ......................................................................................... 40
   8.2. CONFIGURING KDUMP MEMORY USAGE AND TARGET LOCATION IN WEB CONSOLE 40
   8.3. KDUMP USING RHEL SYSTEM ROLES ........................................................... 42
   8.4. ADDITIONAL RESOURCES ........................................................................... 43

**CHAPTER 9. RECOVERING AND RESTORING A SYSTEM** ........................................ 44
9.1. SETTING UP REAR
9.2. USING A REAR RESCUE IMAGE ON THE 64-BIT IBM Z ARCHITECTURE

CHAPTER 10. TROUBLESHOOTING PROBLEMS USING LOG FILES ........................................... 48
  10.1. SERVICES HANDLING SYSLOG MESSAGES 48
  10.2. SUBDIRECTORIES STORING SYSLOG MESSAGES 48
  10.3. INSPECTING LOG FILES USING THE WEB CONSOLE 48
  10.4. VIEWING LOGS USING THE COMMAND LINE 49
  10.5. ADDITIONAL RESOURCES 50

CHAPTER 11. ACCESSING THE RED HAT SUPPORT ........................................................................ 51
  11.1. OBTAINING RED HAT SUPPORT THROUGH RED HAT CUSTOMER PORTAL 51
  11.2. TROUBLESHOOTING PROBLEMS USING SOSREPORT 51

CHAPTER 12. INTRODUCTION TO SYSTEMD ............................................................................. 53
  12.1. SYSTEMD UNIT TYPES 53
  12.2. SYSTEMD MAIN FEATURES 54
  12.3. COMPATIBILITY CHANGES 55

CHAPTER 13. MANAGING SYSTEM SERVICES WITH SYSTEMCTL ........................................... 57
  13.1. SERVICE UNIT MANAGEMENT WITH SYSTEMCTL 57
  13.2. COMPARISON OF A SERVICE UTILITY WITH SYSTEMCTL 57
  13.3. LISTING SYSTEM SERVICES 58
  13.4. Displaying System Service Status 59
  13.5. POSITIVE AND NEGATIVE SERVICE DEPENDENCIES 61
  13.6. STARTING A SYSTEM SERVICE 62
  13.7. STOPPING A SYSTEM SERVICE 62
  13.8. RESTARTING A SYSTEM SERVICE 63
  13.9. ENABLING A SYSTEM SERVICE 64
  13.10. DISABLING A SYSTEM SERVICE 65

CHAPTER 14. WORKING WITH SYSTEMD TARGETS ................................................................. 67
  14.1. DIFFERENCE BETWEEN SYSV RUNLEVELS AND SYSTEMD TARGETS 67
  14.2. VIEWING THE DEFAULT TARGET 68
    14.2.1. Changing the default target 69
    14.2.2. Changing the default target using symbolic link 69
    14.2.2.1. Booting to rescue mode 70
    14.2.2.2. Booting to emergency mode 71

CHAPTER 15. SHUTTING DOWN, SUSPENDING, AND HIBERNATING THE SYSTEM .................. 72
  15.1. SYSTEM SHUTDOWN 72
  15.2. SHUTTING DOWN THE SYSTEM USING THE SHUTDOWN COMMAND 72
  15.3. SHUTTING DOWN THE SYSTEM USING THE SYSTEMCTL COMMAND 73
  15.4. RESTARTING THE SYSTEM 73
  15.5. SUSPENDING THE SYSTEM 74
  15.6. HIBERNATING THE SYSTEM 74
  15.7. OVERVIEW OF THE POWER MANAGEMENT COMMANDS WITH SYSTEMCTL 75

CHAPTER 16. WORKING WITH SYSTEMD UNIT FILES ........................................................... 76
  16.1. INTRODUCTION TO UNIT FILES 76
  16.2. UNIT FILE STRUCTURE 76
  16.3. IMPORTANT [UNIT] SECTION OPTIONS 77
  16.4. IMPORTANT [SERVICE] SECTION OPTIONS 78
  16.5. IMPORTANT [INSTALL] SECTION OPTIONS 79
  16.6. CREATING CUSTOM UNIT FILES 79
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>23.1</td>
<td>Changing the root password as the root user</td>
<td>118</td>
</tr>
<tr>
<td>23.2</td>
<td>Changing or resetting the forgotten root password as a non-root user</td>
<td>118</td>
</tr>
<tr>
<td>23.3</td>
<td>Resetting the root password on boot</td>
<td>118</td>
</tr>
<tr>
<td>24.1</td>
<td>Managing file permissions</td>
<td>121</td>
</tr>
<tr>
<td>24.2</td>
<td>User file-creation mode mask</td>
<td>123</td>
</tr>
<tr>
<td>24.3</td>
<td>Default file permissions</td>
<td>124</td>
</tr>
<tr>
<td>24.4</td>
<td>Changing file permissions using symbolic values</td>
<td>126</td>
</tr>
<tr>
<td>24.5</td>
<td>Changing file permissions using octal values</td>
<td>128</td>
</tr>
<tr>
<td>24.6</td>
<td>Resetting the root password on boot</td>
<td>129</td>
</tr>
<tr>
<td>25.1</td>
<td>Displaying the current value of the umask</td>
<td>129</td>
</tr>
<tr>
<td>25.2</td>
<td>Displaying the default bash umask</td>
<td>129</td>
</tr>
<tr>
<td>25.3</td>
<td>Setting the umask using symbolic values</td>
<td>130</td>
</tr>
<tr>
<td>25.4</td>
<td>Setting the umask using octal values</td>
<td>131</td>
</tr>
<tr>
<td>25.5</td>
<td>Changing the default umask for the non-login shell</td>
<td>131</td>
</tr>
<tr>
<td>25.6</td>
<td>Changing the default umask for the login shell</td>
<td>132</td>
</tr>
<tr>
<td>25.7</td>
<td>Changing the default umask for a specific user</td>
<td>132</td>
</tr>
<tr>
<td>25.8</td>
<td>Setting default umask for newly created home directories</td>
<td>132</td>
</tr>
<tr>
<td>26.1</td>
<td>Managing the access control list</td>
<td>134</td>
</tr>
<tr>
<td>26.2</td>
<td>Setting the access control list</td>
<td>134</td>
</tr>
<tr>
<td>27.1</td>
<td>Using the chrony suite to configure ntp</td>
<td>136</td>
</tr>
<tr>
<td>27.2</td>
<td>Using chronyc to control chronyd</td>
<td>136</td>
</tr>
<tr>
<td>28.1</td>
<td>Managing chrony</td>
<td>138</td>
</tr>
<tr>
<td>28.2</td>
<td>Checking if chrony is synchronized</td>
<td>138</td>
</tr>
<tr>
<td>28.3</td>
<td>Manually adjusting the system clock</td>
<td>139</td>
</tr>
<tr>
<td>28.4</td>
<td>Setting up chrony for a system in an isolated network</td>
<td>140</td>
</tr>
<tr>
<td>28.5</td>
<td>Configuring remote monitoring access</td>
<td>141</td>
</tr>
<tr>
<td>28.6</td>
<td>Managing time synchronization using RHel system roles</td>
<td>142</td>
</tr>
<tr>
<td>28.7</td>
<td>Additional resources</td>
<td>143</td>
</tr>
<tr>
<td>29.1</td>
<td>Chrony with hw timestamping</td>
<td>144</td>
</tr>
<tr>
<td>29.2</td>
<td>Verifying support for hardware timestamping</td>
<td>144</td>
</tr>
<tr>
<td>29.3</td>
<td>Enabling hardware timestamping</td>
<td>145</td>
</tr>
<tr>
<td>29.4</td>
<td>Enabling interleaved mode</td>
<td>145</td>
</tr>
<tr>
<td>29.5</td>
<td>Configuring server for large number of clients</td>
<td>146</td>
</tr>
<tr>
<td>29.6</td>
<td>Verifying hardware timestamping</td>
<td>146</td>
</tr>
<tr>
<td>29.7</td>
<td>Configuring PTP-NTP bridge</td>
<td>147</td>
</tr>
<tr>
<td>30.1</td>
<td>Overview of network time security (NTS) in chrony</td>
<td>148</td>
</tr>
<tr>
<td>30.2</td>
<td>Enabling network time security (NTS) on the server</td>
<td>149</td>
</tr>
<tr>
<td>31.1</td>
<td>Using secure communications between two systems with openssh</td>
<td>151</td>
</tr>
<tr>
<td>31.2</td>
<td>SSH and openssh</td>
<td>151</td>
</tr>
<tr>
<td>31.3</td>
<td>Configuring and starting an openssh server</td>
<td>154</td>
</tr>
</tbody>
</table>

**Note:** The page numbers are placeholders and may not correspond to actual page numbers in the book.
### 31. GENERATING SSH KEY PAIRS

- 31.4. Generating SSH Key Pairs
- 31.5. Using SSH Keys Stored on a Smart Card
- 31.6. Making OpenSSH More Secure
- 31.7. Connecting to a Remote Server Using an SSH Jump Host
- 31.8. Connecting to Remote Machines With SSH Keys Using SSH-Agent
- 31.9. Additional Resources

### 32. Configuring a Remote Logging Solution

- 32.1. The Rsyslog Logging Service
- 32.2. Installing Rsyslog Documentation
- 32.3. Configuring a Server for Remote Logging Over TCP
- 32.4. Configuring Remote Logging to a Server Over TCP
- 32.5. Configuring TLS-Encrypted Remote Logging
- 32.6. Configuring a Server for Receiving Remote Logging Information Over UDP
- 32.7. Configuring Remote Logging to a Server Over UDP
- 32.8. Load Balancing Helper in Rsyslog
- 32.9. Configuring Reliable Remote Logging
- 32.10. Supported Rsyslog Modules
- 32.11. Additional Resources

### 33. Using the Logging System Role

- 33.1. The Logging System Role
- 33.2. Logging System Role Parameters
- 33.3. Applying a Local Logging System Role
- 33.4. Filtering Logs in a Local Logging System Role
- 33.5. Applying a Remote Logging Solution Using the Logging System Role
- 33.6. Using the Logging System Role With TLS
  - 33.6.1. Configuring client logging with TLS
  - 33.6.2. Configuring server logging with TLS
- 33.7. Using the Logging System Roles With RELP
  - 33.7.1. Configuring client logging with RELP
  - 33.7.2. Configuring server logging with RELP
- 33.8. Additional Resources
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.

- 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 feedback via Bugzilla, create a new 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 9, the interface currently consists of the following roles:

- Certificate Issuance and Renewal
- Kernel Settings
- Metrics
- Network Bound Disk Encryption client and Network Bound Disk Encryption server
- Networking
- Postfix
- SSH client
- SSH server
- System-wide Cryptographic Policies
- Terminal Session Recording

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
- Documentation in the `/usr/share/doc/rhel-system-roles/` directory [1]

1.2. RHEL SYSTEM ROLES TERMINOLOGY

You can find the following terms across this documentation:

**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:

  ```bash
  # dnf install rhel-system-roles
  ```

  1. Install the Ansible Core package:

  ```bash
  # dnf install ansible-core
  ```

  The Ansible Core package provides the `ansible-playbook` CLI, the Ansible Vault functionality, and the basic modules and filters required by RHEL Ansible content.

- 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`:

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

   ```bash
   # dnf install rhel-system-roles
   ```
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.postfix
   ```

   **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 `postfix`, `metrics`, `network`, `tlog`, or `ssh`.

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.postfix

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 tool

1.4. ADDITIONAL RESOURCES

- Red Hat Enterprise Linux (RHEL) System Roles
- 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 and later versions use 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:
  
  ```bash
  $ localectl list-locales
  C.utf8
  aa_DJ
  aa_DJ.iso88591
  aa_DJ.utf8
  ...
  ```

- To display the current status of the system locales settings:
  
  ```bash
  $ localectl status
  ```

- To set or change the default system locale settings, use a `localectl set-locale` sub-command as the `root` user. For example:
  
  ```bash
  # 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:
  
  ```bash
  $ localectl list-keymaps
  ANSI-dvorak
  al
  al-plisi
  amiga-de
  amiga-us
  ...
  ```

- To display the current status of keymaps settings:
  
  ```bash
  $ 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.

![GNOME Control Center](image)

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

3. Click the Language menu.

![Language Menu](image)

4. Select the required region and language from the menu.
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 (deprecated), 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:

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

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

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

5. Set the IPv4 and IPv6 default gateways:

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

6. Set the IPv4 and IPv6 DNS server addresses:

```bash
# 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:

```bash
# 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:

```bash
# 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:

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

2. To display all settings of the connection profile:

```bash
# nmcli connection show Example-Connection
connection.id:     Example-Connection
connection.uuid:   b6c6fa1c-e4ad-46e5-a8f8-b75f06b79f76
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:db8:1::2
```

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::fff3
```

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**
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.
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
enp1s0 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: 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)
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

```yaml
# 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

- Getting started with 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:

```bash
# 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:

```bash
# 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 with account credentials 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. Log in to the RHEL web console. For details, see Logging in to the web console.

2. In the Health field in the Overview page, click the Not registered warning, or click Subscriptions in the main menu to move to page with your subscription information.

3. In the Overview filed, click Register.
4. In the Register system dialog box, select that you want to register using your account credentials.

5. Enter your username.

6. Enter your password.

7. Optionally, enter your organization’s name or ID.
   If your account belongs to more than one organization 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.
   - If you do not want to connect your system to Red Hat Insights, uncheck the Insights check box.

8. Click the Register button.

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

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
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:

   ```shell
   # 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:

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

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

   ```shell
   # 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:

   ```shell
   # 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 `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 Toggle button.
   d. To stop the service, click the button and choose the option **Stop**.
CHAPTER 5. MAKING SYSTEMD SERVICES START AT Boot TIME

**NFS client services**

**Status:** Running

**Path:** /etc/systemd/system/nfs-c

**Wants:** rpc.statd.notify, service, auth_rpcgs-module.service, remote-fs-pre.target

**Wanted By:** remote-fs.target, multi-user.target

**Conflicts:** shutdown.target

**Before:** remote-fs-pre.target, shutdown.target, multi-user.target

**After:** rpc-gssd.service, gssproxy.service

**Service Logs**

March 29, 2020

1:03 PM Reached target NFS client services.
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:

  ```bash
  $ 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:

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

**Verification steps**

- Check that firewalld is running and enabled:

  ```bash
  $ 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 firewalld
- man firewalld(1)
6.2. 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

6.3. 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:

   ```bash
   $ 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:

```shell
# 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. For more information, see Changing SELinux modes at boot time.

Additional resources

- Changing SELinux states and modes

6.4. ADDITIONAL RESOURCES

- Generating SSH key pairs
- Setting an OpenSSH server for key-based authentication
- Security hardening
- Using SELinux
- Securing networks
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.

![Example User](Image)

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 which provides a crash dumping mechanism. The service enables you to save the contents of the system memory for 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 into a file. 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, operational `kdump` 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.

You can enable `kdump` for all installed kernels on a machine or only for specified kernels. This is useful when there are multiple kernels used on a machine, some of which are stable enough that there is no concern that they could crash.

When `kdump` is installed, a default `/etc/kdump.conf` file is created. The file includes the default minimum `kdump` configuration. You can edit this file to customize the `kdump` configuration, but it is not required.

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.

5. Test your configuration by crashing the kernel.
a. Click **Test configuration**.

b. In the **Test kdump settings** field, click **Crash system**.

---

**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:

```yaml
---
- 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.
3. Create rescue system.
4. Generate backup files.

9.1. SETTING UP REAR

Use the following steps to install the package 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 the ReaR utility by running the following command:

   # dnf install rear

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

   # vi /etc/rear/local.conf

3. 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
4. To configure ReaR to keep the previous backup archive when the new one is created, also add the following line to the configuration file:

```
NETFS_KEEP_OLD_BACKUP_COPY=y
```

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

```
BACKUP_TYPE=incremental
```

6. Create a rescue system:

```
# rear mkrescue
```

7. Take a backup as per the restore plan. For example, in the case of the `NETFS` backup method, run the following command:

```
# rear mkbackuponly
```

Alternatively, you can create the rescue system and the backup in a single step by running the following command:

```
# rear mkbackup
```

This command combines the functionality of the `rear mkrescue` and `rear mkbackuponly` commands.

## 9.2. USING A REAR RESCUE IMAGE ON THE 64-BIT IBM Z ARCHITECTURE

Basic Relax and Recover (ReaR) functionality is now available on the 64-bit IBM Z architecture as a Technology Preview. You can create a ReaR rescue image on IBM Z only in the z/VM environment. Backing up and recovering logical partitions (LPARs) has not been tested.

### IMPORTANT

ReaR on the 64-bit IBM Z architecture is a Technology Preview feature only. Technology Preview features are not supported with Red Hat production service level agreements (SLAs) and might not be functionally complete. Red Hat does not recommend using them in production. These features provide early access to upcoming product features, enabling customers to test functionality and provide feedback during the development process. For more information about the support scope of Red Hat Technology Preview features, see [https://access.redhat.com/support/offerings/techpreview](https://access.redhat.com/support/offerings/techpreview).

The only output method currently available is Initial Program Load (IPL). IPL produces a kernel and an initial ramdisk (initrd) that can be used with the zIPL bootloader.

### Prerequisites
ReaR is installed.

- To install ReaR, run the command `yum install rear`

**Procedure**

Add the following variables to the `/etc/rear/local.conf` to configure ReaR for producing a rescue image on the 64-bit IBM Z architecture:

1. To configure the IPL output method, add `OUTPUT=IPL`.

2. To configure the backup method and destination, add `BACKUP` and `BACKUP_URL` variables. For example:

   ```
   BACKUP=NETFS
   BACKUP_URL=nfs://<nfsserver name>/<share path>
   ```

   **IMPORTANT**

   The local backup storage is currently not supported on the 64-bit IBM Z architecture.

3. Optionally, you can also configure the `OUTPUT_URL` variable to save the kernel and `initrd` files. By default, the `OUTPUT_URL` is aligned with `BACKUP_URL`.

4. To perform backup and rescue image creation:

   ```
   rear mkbackup
   ```

5. This creates the kernel and initrd files at the location specified by the `BACKUP_URL` or `OUTPUT_URL` (if set) variable, and a backup using the specified backup method.

6. To recover the system, use the ReaR kernel and initrd files created in step 3, and boot from a Direct Attached Storage Device (DASD) or a Fibre Channel Protocol (FCP)-attached SCSI device prepared with the `zipl` boot loader, kernel, and `initrd`. For more information, see Using a Prepared DASD.

7. When the rescue kernel and `initrd` get booted, it starts the ReaR rescue environment. Proceed with system recovery.

**WARNING**

Currently, the rescue process reformats all the DASDs (Direct Attached Storage Devices) connected to the system. Do not attempt a system recovery if there is any valuable data present on the system storage devices. This also includes the device prepared with the `zipl` bootloader, ReaR kernel, and `initrd` that were used to boot into the rescue environment. Ensure to keep a copy.

**Additional resources**
• Installing under z/VM
• Recovering and restoring a system
• Using a Prepared DASD
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:

```shell
$ 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

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>journalctl</code></td>
<td>Shows all collected journal entries.</td>
</tr>
<tr>
<td><code>journalctl FILEPATH</code></td>
<td>Shows logs related to a specific file. For example, the <code>journalctl /dev/sda</code> command displays logs related to the <code>/dev/sda</code> file system.</td>
</tr>
<tr>
<td><code>journalctl -b</code></td>
<td>Shows logs for the current boot.</td>
</tr>
<tr>
<td><code>journalctl -k -b -1</code></td>
<td>Shows kernel logs for the current boot.</td>
</tr>
</tbody>
</table>

### Table 10.2. Viewing information on specific services
### Command

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

#### Table 10.3. Viewing logs related to specific boots

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>journalctl --list-boots</code></td>
<td>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.</td>
</tr>
<tr>
<td><code>journalctl --boot=ID _SYSTEMD_UNIT=foo</code></td>
<td>Shows information about the specified boot ID.</td>
</tr>
</tbody>
</table>

### 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:

   ```
   # dnf 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 an sosreport and how to create one in Red Hat Enterprise Linux?
CHAPTER 12. 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 12.1. systemd unit files locations

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

The units encapsulate information about:

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

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_
```

12.1. SYSTEMD UNIT TYPES

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

Table 12.2. Available systemd unit types

<table>
<thead>
<tr>
<th>Unit Type</th>
<th>File Extension</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Service unit</td>
<td><code>.service</code></td>
<td>A system service.</td>
</tr>
</tbody>
</table>
### 12.2. 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.

12.3. 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](Changing the timeout limit).
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.

### 13.1. 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

### 13.2. COMPARISON OF A SERVICE UTILITY WITH SYSTEMCTL

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

Table 13.1. Comparison of the service utility with systemctl
13.3. 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:

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

UNIT                     LOAD   ACTIVE SUB     DESCRIPTION
abrt-ccpp.service        loaded active exited  Install ABRT coredump hook
abrt-oops.service       loaded active running ABRT kernel log watcher
abrd病.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
  ```

<table>
<thead>
<tr>
<th>UNIT FILE</th>
<th>STATE</th>
</tr>
</thead>
<tbody>
<tr>
<td>abrt-ccpp.service</td>
<td>enabled</td>
</tr>
<tr>
<td>abrt-oops.service</td>
<td>enabled</td>
</tr>
<tr>
<td>abrtd.service</td>
<td>enabled</td>
</tr>
<tr>
<td>...</td>
<td></td>
</tr>
<tr>
<td>wpa_supplicant.service</td>
<td>disabled</td>
</tr>
<tr>
<td>ypbind.service</td>
<td>disabled</td>
</tr>
</tbody>
</table>

  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](#)

## 13.4. 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 13.2. Available service unit information

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

Example 13.1. 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/gnome...

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:

  ```bash
  # 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:

  ```bash
  # 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:

  ```bash
  # 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:

  ```bash
  # 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
  └─ shutdown.target
  ```

Additonal resources

- Listing system services

13.5. 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

### 13.6. 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 13.2. 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

### 13.7. 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 13.3. 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

### 13.8. 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 13.4. 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

### 13.9. 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 reenable <name>.service
  ```

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

**Example 13.5. 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

### 13.10. 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 subdirectories.

**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`:

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

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

  **Example 13.6. 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`:

  ```bash
  # 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`:

  ```bash
  # 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:

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

**Additional resources**
- Displaying system service status
• Stopping a system service
CHAPTER 14. 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.

14.1. 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 14.1. Comparison of SysV runlevels with systemd targets

<table>
<thead>
<tr>
<th>Runlevel</th>
<th>Target Units</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>runlevel0.target, poweroff.target</td>
<td>Shut down and power off the system.</td>
</tr>
<tr>
<td>1</td>
<td>runlevel1.target, rescue.target</td>
<td>Set up a rescue shell.</td>
</tr>
<tr>
<td>2</td>
<td>runlevel2.target, multi-user.target</td>
<td>Set up a non-graphical multi-user system.</td>
</tr>
<tr>
<td>3</td>
<td>runlevel3.target, multi-user.target</td>
<td>Set up a non-graphical multi-user system.</td>
</tr>
<tr>
<td>4</td>
<td>runlevel4.target, multi-user.target</td>
<td>Set up a non-graphical multi-user system.</td>
</tr>
<tr>
<td>5</td>
<td>runlevel5.target, graphical.target</td>
<td>Set up a graphical multi-user system.</td>
</tr>
<tr>
<td>6</td>
<td>runlevel6.target, reboot.target</td>
<td>Shut down and reboot the system.</td>
</tr>
</tbody>
</table>
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>
<thead>
<tr>
<th>Table 14.2. Comparison of SysV init commands with systemctl</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Old Command</strong></td>
</tr>
<tr>
<td>runlevel</td>
</tr>
<tr>
<td>telinit runlevel</td>
</tr>
</tbody>
</table>

---

**Additional resources**

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

---

### 14.2. 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:
  ```bash
  $ systemctl get-default
  graphical.target
  ```
- To determine the default target using the symbolic link:
  ```bash
  $ 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:
  ```bash
  $ systemctl list-units --type target --all
  ```
- To list all currently loaded target units:
$ systemctl list-units --type target

<table>
<thead>
<tr>
<th>UNIT</th>
<th>LOAD</th>
<th>ACTIVE</th>
<th>SUB</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>basic.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Basic System</td>
</tr>
<tr>
<td>cryptsetup.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Encrypted Volumes</td>
</tr>
<tr>
<td>getty.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Login Prompts</td>
</tr>
<tr>
<td>graphical.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Graphical Interface</td>
</tr>
<tr>
<td>local-fs-pre.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Local File Systems (Pre)</td>
</tr>
<tr>
<td>local-fs.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Local File Systems</td>
</tr>
<tr>
<td>multi-user.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Multi-User System</td>
</tr>
<tr>
<td>network.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Network</td>
</tr>
<tr>
<td>paths.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Paths</td>
</tr>
<tr>
<td>remote-fs.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Remote File Systems</td>
</tr>
<tr>
<td>sockets.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Sockets</td>
</tr>
<tr>
<td>sound.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Sound Card</td>
</tr>
<tr>
<td>spice-vdagentd.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Agent daemon for Spice guests</td>
</tr>
<tr>
<td>swap.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Swap</td>
</tr>
<tr>
<td>sysinit.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>System Initialization</td>
</tr>
<tr>
<td>time-sync.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>System Time Synchronized</td>
</tr>
<tr>
<td>timers.target</td>
<td>loaded</td>
<td>active</td>
<td>active</td>
<td>Timers</td>
</tr>
</tbody>
</table>

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.

14.2.1. 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

14.2.2. 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
  ```

### 14.2.2.1. 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:

  ```bash
  # 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:

  ```bash
  # systemctl --no-wall rescue
  ```

14.2.2.2. 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:

  ```bash
  # 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:

  ```bash
  # systemctl --no-wall emergency
  ```
CHAPTER 15. SHUTTING DOWN, SUSPENDING, AND HIBERNATING THE SYSTEM

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

15.1. 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`

15.2. 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

### 15.3. 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

### 15.4. 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:

```bash
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.

### 15.5. 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:

```bash
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

### 15.6. 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

15.7. 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 15.1. Overview of the systemctl power management commands

<table>
<thead>
<tr>
<th>systemctl command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>systemctl halt</td>
<td>Halts the system.</td>
</tr>
<tr>
<td>systemctl poweroff</td>
<td>Powers off the system.</td>
</tr>
<tr>
<td>systemctl reboot</td>
<td>Restarts the system.</td>
</tr>
<tr>
<td>systemctl suspend</td>
<td>Suspends the system.</td>
</tr>
<tr>
<td>systemctl hibernate</td>
<td>Hibernates the system.</td>
</tr>
<tr>
<td>systemctl hybrid-sleep</td>
<td>Hibernates and suspends the system.</td>
</tr>
</tbody>
</table>
CHAPTER 16. 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

16.1. 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.

16.2. 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

#### 16.3. IMPORTANT **[UNIT]** SECTION OPTIONS

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

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

[a] For a complete list of options configurable in the **[Unit]** section, see the `systemd.unit(5)` manual page.

[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.
## 16.4. IMPORTANT [SERVICE] SECTION OPTIONS

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

### Table 16.2. Important [Service] section options

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
</table>
| **Type** | Configures the unit process startup type that affects the functionality of **ExecStart** and related options. One of:  
  * 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 sd_notify() 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** | 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. |
| **ExecStop** | Specifies commands or scripts to be executed when the unit is stopped. |
| **ExecReload** | Specifies commands or scripts to be executed when the unit is reloaded. |
| **Restart** | With this option enabled, the service is restarted after its process exits, with the exception of a clean stop by the **systemctl** command. |
16.5. IMPORTANT [INSTALL] SECTION OPTIONS

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

Table 16.3. Important [Install] section options

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alias</td>
<td>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.</td>
</tr>
<tr>
<td>RequiredBy</td>
<td>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.</td>
</tr>
<tr>
<td>WantedBy</td>
<td>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.</td>
</tr>
<tr>
<td>Also</td>
<td>Specifies a list of units to be installed or uninstalled along with the unit.</td>
</tr>
<tr>
<td>DefaultInstance</td>
<td>Limited to instantiated units, this option specifies the default instance for which the unit is enabled. See Working with instantiated units.</td>
</tr>
</tbody>
</table>

For a complete list of options configurable in the [Install] section, see the systemd.unit(5) manual page.

16.6. 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 custom unit file by using the 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`:

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

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

3. Open the `name.service` file created in the previous step, and add the service configuration options. There is a variety of options that can be used depending on the type of service you wish to create, see Unit file structure.

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

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

   [Service]
   ExecStart=path_to_executable
   Type=forking
   PIDFile=path_to_pidfile

   [Install]
   WantedBy=default.target
   ```

   Where:

   - `service_description` is an informative description that is displayed in journal log files and in the output of the `systemctl status` command.
   - the `After` setting ensures that the service is started only after the network is running. Add a space-separated list of other relevant services or targets.
   - `path_to_executable` stands for the path to the actual service executable.
   - `Type=forking` is used for daemons that make the fork system call. The main process of the service is created with the PID specified in `path_to_pidfile`. Find other startup types in 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.

### 16.7. 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/sshd
ExecStart=/usr/sbin/sshd -D -f /etc/ssh/sshd-second_config $OPTIONS
ExecReload=/bin/kill -HUP $MAINPID
KillMode=process
Restart=on-failure
RestartSec=42s

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

5. If using SELinux, add the port for the second instance of sshd to SSH ports, otherwise the second instance of sshd will be rejected to bind to the port:

```
# semanage port -a -t ssh_port_t -p tcp 22220
```

6. Enable sshd-second.service, so that it starts automatically upon boot:

```
# systemctl enable sshd-second.service
```

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.

**16.8. 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.

### 16.9. 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.

### 16.10. 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 16.4. Dependency options from the LSB header**

<table>
<thead>
<tr>
<th>LSB Option</th>
<th>Description</th>
<th>Unit File Equivalent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Provides</td>
<td>Specifies the boot facility name of the service, that can be referenced in other init scripts (with the &quot;$&quot; prefix). This is no longer needed as unit files refer to other units by their file names.</td>
<td>–</td>
</tr>
</tbody>
</table>
16.11. 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 `multi-user.target` and `graphical.target`. Note that the `graphical.target` depends on `multi-user.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.

16.12. 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.

```bash
conf_check() {
```
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.

### 16.13. 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.
1. Depending on the extent of the required changes, pick one of the following approaches:
   - Create a directory for supplementary configuration files at `/etc/systemd/system/unit.d/`. This method is recommended for most use cases. It enables extending the default configuration with additional functionality, while still referring to the original unit file. Changes to the default unit introduced with a package upgrade are therefore applied automatically. See Extending the default unit configuration for more information.
   - Create a copy of the original unit file `/usr/lib/systemd/system/` in `/etc/systemd/system/` and make changes there. The copy overrides the original file, therefore changes introduced with the package update are not applied. This method is useful for making significant unit changes that should persist regardless of package updates. See Overriding the default unit configuration for details.

2. To return to the default configuration of the unit, delete custom-created configuration files in `/etc/systemd/system/`.

3. 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
   ```

4. If the modified unit file belongs to a running service, this service must be restarted to accept new settings:
   ```
   systemctl restart name.service
   ```

   IMPORTANT

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

   Then manage this service in the same way as a normal `systemd` service.

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

---

16.14. 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 16.1. 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:

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

3. To apply the unit changes, execute:

```plaintext
# 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 systemctl-delta command, described in Monitoring overridden units, such units are always marked as [EXTENDED], even though in sum, certain options are actually overridden.

16.15. 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:

```plaintext
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:

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

16.16. 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.

### 16.17. 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

+ /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
```

---
16.18. 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`.

16.19. 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 16.5. Important unit specifiers

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

For a complete list of unit specifiers, see the \texttt{systemd.unit(5)} manual page.

## 16.20. 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 17. 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.

17.1. 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

```bash
$ 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:

```bash
$ systemd-analyze
```

Analyzing unit initialization time

Procedure

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

```bash
$ 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 17.1. The output of the systemd-analyze critical-chain command

17.2. 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 17.1. Services enabled by default on a minimal installation of RHEL
<table>
<thead>
<tr>
<th>Service name</th>
<th>Can it be disabled?</th>
<th>More information</th>
</tr>
</thead>
<tbody>
<tr>
<td>auditd.service</td>
<td>yes</td>
<td>Disable 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.</td>
</tr>
<tr>
<td>autovt.service</td>
<td>no</td>
<td>This service runs only when it is really needed, so it does not need to be disabled.</td>
</tr>
<tr>
<td>crond.service</td>
<td>yes</td>
<td>Be aware that no items from crontab will run if you disable crond.service.</td>
</tr>
<tr>
<td>dbus-org.fedoraproject.FirewallD1.service</td>
<td>yes</td>
<td>A symlink to firewalld.service</td>
</tr>
<tr>
<td>dbus-org.freedesktop.NetworkManager.service</td>
<td>yes</td>
<td>A symlink to NetworkManager.service</td>
</tr>
<tr>
<td>dbus-org.freedesktop.Nm-dispatcher.service</td>
<td>yes</td>
<td>A symlink to NetworkManager-dispatcher.service</td>
</tr>
<tr>
<td>firewalld.service</td>
<td>yes</td>
<td>Disable firewalld.service only if you do not need firewall.</td>
</tr>
<tr>
<td><a href="mailto:getty@.service">getty@.service</a></td>
<td>no</td>
<td>This service runs only when it is really needed, so it does not need to be disabled.</td>
</tr>
<tr>
<td>import-state.service</td>
<td>yes</td>
<td>Disable import-state.service only if you do not need to boot from a network storage.</td>
</tr>
<tr>
<td>irqbalance.service</td>
<td>yes</td>
<td>Disable irqbalance.service only if you have just one CPU. Do not disable irqbalance.service on systems with multiple CPUs.</td>
</tr>
<tr>
<td>Service name</td>
<td>Can it be disabled?</td>
<td>More information</td>
</tr>
<tr>
<td>---------------------------</td>
<td>---------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>kdump.service</td>
<td>yes</td>
<td>Disable <strong>kdump.service</strong> only if you do not need reports from kernel crashes.</td>
</tr>
<tr>
<td>loadmodules.service</td>
<td>yes</td>
<td>This service is not started unless the <code>/etc/rc.modules</code> or <code>/etc/sysconfig/modules</code> directory exists, which means that it is not started on a minimal RHEL installation.</td>
</tr>
<tr>
<td>lvm2-monitor.service</td>
<td>yes</td>
<td>Disable <strong>lvm2-monitor.service</strong> only if you do not use Logical Volume Manager (LVM).</td>
</tr>
<tr>
<td>microcode.service</td>
<td>no</td>
<td>Do not disable the service because it provides updates of the microcode software in CPU.</td>
</tr>
<tr>
<td>NetworkManager-dispatcher.service</td>
<td>yes</td>
<td>Disable <strong>NetworkManager-dispatcher.service</strong> only if you do not need notifications on network configuration changes (for example in static networks).</td>
</tr>
<tr>
<td>NetworkManager-wait-online.service</td>
<td>yes</td>
<td>Disable <strong>NetworkManager-wait-online.service</strong> only if you do not need working network connection available right after the boot. If the service is enabled, the system does not finish the boot before the network connection is working. This may prolong the boot time significantly.</td>
</tr>
<tr>
<td>NetworkManager.service</td>
<td>yes</td>
<td>Disable <strong>NetworkManager.service</strong> only if you do not need connection to a network.</td>
</tr>
<tr>
<td>nis-domainname.service</td>
<td>yes</td>
<td>Disable <strong>nis-domainname.service</strong> only if you do not use Network Information Service (NIS).</td>
</tr>
<tr>
<td>rhsmcertd.service</td>
<td>no</td>
<td></td>
</tr>
<tr>
<td>rngd.service</td>
<td>yes</td>
<td>Disable <strong>rngd.service</strong> only if you do not need a lot of entropy on your system, or you do not have any sort of hardware generator. Note that the service is necessary in environments that require a lot of good entropy, such as systems used for generation of X.509 certificates (for example the FreeIPA server).</td>
</tr>
<tr>
<td>rsyslog.service</td>
<td>yes</td>
<td>Disable <strong>rsyslog.service</strong> only if you do not need persistent logs, or you set <strong>systemd-journald</strong> to persistent mode.</td>
</tr>
<tr>
<td>Service name</td>
<td>Can it be disabled</td>
<td>More information</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>-------------------</td>
<td>----------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>selinux-autorelabel-mark.service</td>
<td>yes</td>
<td>Disable <code>selinux-autorelabel-mark.service</code> only if you do not use SELinux.</td>
</tr>
<tr>
<td>sshd.service</td>
<td>yes</td>
<td>Disable <code>sshd.service</code> only if you do not need remote logins by OpenSSH server.</td>
</tr>
<tr>
<td>sssd.service</td>
<td>yes</td>
<td>Disable <code>sssd.service</code> 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 <code>sssd-*</code> units if you disable <code>sssd.service</code>.</td>
</tr>
<tr>
<td>syslog.service</td>
<td>yes</td>
<td>An alias for <code>rsyslog.service</code></td>
</tr>
<tr>
<td>tuned.service</td>
<td>yes</td>
<td>Disable <code>tuned.service</code> only if you do need to use performance tuning.</td>
</tr>
<tr>
<td>lvm2-lvmpolld.socket</td>
<td>yes</td>
<td>Disable <code>lvm2-lvmpolld.socket</code> only if you do not use Logical Volume Manager (LVM).</td>
</tr>
<tr>
<td>dnf-makecache.timer</td>
<td>yes</td>
<td>Disable <code>dnf-makecache.timer</code> only if you do not need your package metadata to be updated automatically.</td>
</tr>
<tr>
<td>unbound-anchor.timer</td>
<td>yes</td>
<td>Disable <code>unbound-anchor.timer</code> only if you do not need daily update of the root trust anchor for DNS Security Extensions (DNSSEC). This root trust anchor is used by Unbound resolver and resolver library for DNSSEC validation.</td>
</tr>
</tbody>
</table>

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

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

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.
17.3. ADDITIONAL RESOURCES

- systemctl(1) man page
- systemctl(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 18. 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.

18.1. 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.

18.2. 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:

```bash
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.

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

3. Modify the UID_MIN value to start at 5000.

```bash
# 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.

### 18.3. 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 19. 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.

19.1. 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.

19.2. 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.

19.3. 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 9 web console.

2. Click Accounts.
3. Select the user account for which to enforce password expiration.

4. In the user account settings, click the second edit.

5. In the Password Expiration dialog box, select Require password change every ... days and enter a positive whole number representing the number of days after which the password expires.

6. Click Change.

Verification steps

- To verify that the password expiration is set, open the account settings. The RHEL 9 web console displays a link with the date of expiration.

---

19.4. 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 9 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 9 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 20. 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.

20.1. 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 20.1. 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

20.2. 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 20.2. 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

---

**20.3. 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 20.3. 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
```

20.4. 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+rwxs 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 21. 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.

21.1. 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.

21.2. 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 21.1. 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 21.2. 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.

### 21.3. 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 21.3. 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
```
- Verify that you changed the primary group of the user:

```bash
$ groups sarah
```

The output displays:

```
sarah : sarah2
```

### 21.4. 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:

```bash
# 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 21.4. Adding a user to a supplementary group**

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

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

**Verification steps**

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

```bash
# groups sysadmin
```

The output displays:

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

### 21.5. 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**
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 21.5. 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
```

**21.6. 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 21.6. 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 22. 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.

22.1. 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 `~`.

### 22.2. 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)
```

22.3. 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 create and edit file in the sudoers.d directory.

**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/dnf*).

4. Save any changes, and exit the editor.

**Example 22.1. Enabling an unprivileged user to install programs with dnf**

To enable the user *sarah* to install programs on the *localhost.localdomain* workstation using the *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/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 *dnf* command with *sudo* privileges, switch the account:

   ```
   # su sarah -
   ```

6. Enter the *sudo dnf* command:

   ```
   $ sudo dnf
   [sudo] password for sarah:
   ```

   Enter the *sudo* password for the user *sarah*.

7. The system displays the list of *dnf* commands and options:

   ```
   ... usage: dnf [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.
22.4. ADDITIONAL RESOURCES

- The `sudo(8)` man page
- The `visudo(8)` man page
CHAPTER 23. 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.

23.1. 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:

  ```bash
  # passwd
  ```

  You are prompted to enter your current password before you can change it.

23.2. 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:

  ```bash
  $ sudo passwd root
  ```

  You are prompted to enter your current non-root password before you can change the root password.

23.3. 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 gfxpayload=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:

```
pwd
```

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 24. 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 \( (umask) \).

24.1. 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 \textit{symbolic} or \textit{octal} values.

<table>
<thead>
<tr>
<th>Permission</th>
<th>Symbolic value</th>
<th>Octal value</th>
</tr>
</thead>
<tbody>
<tr>
<td>No permission</td>
<td>---</td>
<td>0</td>
</tr>
<tr>
<td>Execute</td>
<td>--x</td>
<td>1</td>
</tr>
<tr>
<td>Write</td>
<td>-w-</td>
<td>2</td>
</tr>
<tr>
<td>Write and execute</td>
<td>-wx</td>
<td>3</td>
</tr>
<tr>
<td>Read</td>
<td>r--</td>
<td>4</td>
</tr>
<tr>
<td>Read and execute</td>
<td>r-x</td>
<td>5</td>
</tr>
<tr>
<td>Read and write</td>
<td>rw-</td>
<td>6</td>
</tr>
</tbody>
</table>
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 24.1. 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 24.2. Permissions for a directory**

If a directory has the following permissions:

```
$ ls -dl
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.

### 24.2. 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.

<table>
<thead>
<tr>
<th>Permission</th>
<th>Symbolic value</th>
<th>Octal value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Read, write, and execute</td>
<td>rwx</td>
<td>0</td>
</tr>
<tr>
<td>Read and write</td>
<td>rw-</td>
<td>1</td>
</tr>
<tr>
<td>Read and execute</td>
<td>r-x</td>
<td>2</td>
</tr>
<tr>
<td>Read</td>
<td>r--</td>
<td>3</td>
</tr>
<tr>
<td>Write and execute</td>
<td>-wx</td>
<td>4</td>
</tr>
<tr>
<td>Write</td>
<td>-w-</td>
<td>5</td>
</tr>
<tr>
<td>Execute</td>
<td>--x</td>
<td>6</td>
</tr>
<tr>
<td>No permissions</td>
<td>---</td>
<td>7</td>
</tr>
</tbody>
</table>

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 24.3. 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**.
24.3. 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 24.4. 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).

<table>
<thead>
<tr>
<th></th>
<th>Symbolic value</th>
<th>Octal value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base permission</td>
<td>rwxrwxrwx</td>
<td>777</td>
</tr>
<tr>
<td>Umask</td>
<td>rwxrwxr-x</td>
<td>002</td>
</tr>
<tr>
<td>Default permission</td>
<td>rwxrwxr-x</td>
<td>775</td>
</tr>
</tbody>
</table>

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 24.5. 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-).
### Example 24.6. 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)*.

<table>
<thead>
<tr>
<th>Symbolic value</th>
<th>Octal value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Base permission</strong></td>
<td><strong>rw-rwxrwx</strong></td>
</tr>
<tr>
<td><strong>Umask</strong></td>
<td><strong>rwxr-xr-x</strong></td>
</tr>
<tr>
<td><strong>Default permission</strong></td>
<td><strong>rwxr-xr-x</strong></td>
</tr>
</tbody>
</table>

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 24.7. 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--)*.

<table>
<thead>
<tr>
<th>Symbolic value</th>
<th>Octal value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Base permission</strong></td>
<td><strong>rw-rw-rw-</strong></td>
</tr>
<tr>
<td><strong>Umask</strong></td>
<td><strong>rwxr-xr-x</strong></td>
</tr>
<tr>
<td><strong>Default permission</strong></td>
<td><strong>rw-r--r--</strong></td>
</tr>
</tbody>
</table>

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 (rwxrwxrwx). However, directories can be created with execute permissions.

24.4. 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 (rwx) my-script.sh, use the chmod a=rwx my-script.sh command.

  See Base file permissions for more details.

Verification steps

- To see the permissions for a particular file, use:

  $ ls -l file-name
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 24.8. 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 (`rwx`) 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 and execute (`r-x`) access for all users (`a`):
      ```
      $ chmod o+r x 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
      ```
24.5. 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 file permissions](#) for more details.
You can use the `umask` utility to display, set, or change the current or default value of the `umask`.

### 25.1. 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:
  ```sh
  $ umask -S
  ```
- To display the current value of the `umask` in the octal mode, use:
  ```sh
  $ 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, SUID bit, or SGID bit). If the first digit is set to 0, the special bit is not set.

### 25.2. 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 25.1. 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.
  ```sh
  $ 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.
  ```sh
  # 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
  ```

25.3. 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.

25.4. 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.

25.5. 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:

   ```bash
   if [ $UID -gt 199 ] && [ “id -gn” = “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.

25.6. 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:

```bash
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.

25.7. 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.

  ```bash
  $ 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.

25.8. 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 26. 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).

26.1. 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.

26.2. 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 26.1. 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 27. 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.

27.1. 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.

27.2. 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 \texttt{chronyd} using the command line utility \texttt{chronyc} in interactive mode, enter the following command as \texttt{root}:

\begin{verbatim}
# chronyc
\end{verbatim}

\texttt{chronyc} must run as \texttt{root} if some of the restricted commands are to be used.

The \texttt{chronyc} command prompt will be displayed as follows:

\begin{verbatim}
chronyc>
\end{verbatim}

2. To list all of the commands, type \texttt{help}.

3. Alternatively, the utility can also be invoked in non-interactive command mode if called together with a command as follows:

\begin{verbatim}
chronyc command
\end{verbatim}

\textbf{NOTE}

Changes made using \texttt{chronyc} are not permanent, they will be lost after a \texttt{chronyd} restart. For permanent changes, modify \texttt{/etc/chrony.conf}.
CHAPTER 28. 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.

28.1. 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**:

   ```bash
   # dnf 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:

   ```bash
   $ 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**:

   ```bash
   # systemctl start chronyd
   ```

   To ensure `chronyd` starts automatically at system start, issue the following command as **root**:

   ```bash
   # systemctl enable chronyd
   ```

4. To stop `chronyd`, issue the following command as **root**:

   ```bash
   # systemctl stop chronyd
   ```

   To prevent `chronyd` from starting automatically at system start, issue the following command as **root**:

   ```bash
   # systemctl disable chronyd
   ```

28.2. 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

2. 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.

3. 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

28.3. 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 `rtcfile` directive is used, the real-time clock should not be manually adjusted. Random adjustments would interfere with `chrony`'s need to measure the rate at which the real-time clock drifts.

### 28.4. 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.

### 28.5. CONFIGURING REMOTE MONITORING ACCESS

`chronyc` can access `chronyd` in two ways:

- Internet Protocol, IPv4 or IPv6.
- Unix domain socket, which is accessible locally by the `root` or `chrony` user.

By default, `chronyc` connects to the Unix domain socket. The default path is `/var/run/chrony/chronyd.sock`. If this connection fails, which can happen for example when `chronyc` is running under a non-root user, `chronyc` tries to connect to `127.0.0.1` and then `::1`.

Only the following monitoring commands, which do not affect the behavior of `chronyd`, are allowed from the network:

- `activity`
- `manual list`
- `rtcdata`
- `smoothing`
- `sources`
- `sourcestats`
- `tracking`
- `waitsync`

The set of hosts from which `chronyd` accepts these commands can be configured with the `cmdallow` directive in the configuration file of `chronyd`, or the `cmdallow` command in `chronyc`. By default, the commands are accepted only from localhost (127.0.0.1 or `::1`).

All other commands are allowed only through the Unix domain socket. When sent over the network, `chronyd` responds with a `Not authorised` error, even if it is from localhost.

The following procedure describes how to access `chronyd` remotely with `chronyc`.

**Procedure**

1. Allow access from both IPv4 and IPv6 addresses by adding the following to the `/etc/chrony.conf` file:
bindcmdaddress 0.0.0.0

or

bindcmdaddress ::

2. Allow commands from the remote IP address, network, or subnet by using the `cmdallow` directive.
   Add the following content to the `/etc/chrony.conf` file:

   ```
   cmdallow 192.168.1.0/24
   ```

3. Open port 323 in the firewall to connect from a remote system.

   ```
   # firewall-cmd --zone=public --add-port=323/udp
   ```

   If you want to open port 323 permanently, use the `--permanent`.

   ```
   # firewall-cmd --permanent --zone=public --add-port=323/udp
   ```

Additional resources

- `chrony.conf(5)` man page

## 28.6. MANAGING TIME SYNCHRONIZATION USING RHEL SYSTEM ROLES

You can manage time synchronization on multiple target machines using the `timesync` role. The `timesync` role installs and configures an NTP or PTP implementation to operate as an NTP client or PTP slave in order to synchronize the system clock with NTP servers or grandmasters in PTP domains.

### WARNING

The `timesync` role replaces the configuration of the given or detected provider service on the managed host. Previous settings are lost, even if they are not specified in the role variables. The only preserved setting is the choice of provider if the `timesync_ntp_provider` variable is not defined.

The following example shows how to apply the `timesync` role in a situation with just one pool of servers.

#### Example 28.1. An example playbook applying the timesync role for a single pool of servers

```yaml
---
- hosts: timesync-test
  vars:
    timesync_ntp_servers:
      - hostname: 2.rhel.pool.ntp.org
```
pool: yes
iburst: yes
roles:
  - rhel-system-roles.timesync

For a detailed reference on timesync 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/timesync directory.

Additional resources

- Getting started with RHEL System Roles

28.7. ADDITIONAL RESOURCES

- chronyc(1) man page
- chronyd(8) man page
- Frequently Asked Questions
CHAPTER 29. 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

29.1. 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 29.1. 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:
```
29.2. ENABLING HARDWARE TIMESTAMPING

To enable hardware timestamping, use the \texttt{hwtimestamp} directive in the \texttt{/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 \texttt{ptp4l} from the \texttt{linuxptp} package, is using hardware timestamping on an interface. Multiple \texttt{hwtimestamp} directives are allowed in the chrony configuration file.

\textbf{Example 29.2. Enabling hardware timestamping by using the hwtimestamp directive}

\begin{verbatim}
hwtimestamp eth0
hwtimestamp eth1
hwtimestamp *
\end{verbatim}

29.3. 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 \texttt{/etc/chrony.conf} specifies a local NTP server using one second polling interval:

\begin{verbatim}
server ntp.local minpoll 0 maxpoll 0
\end{verbatim}

29.4. ENABLING INTERLEAVED MODE

NTP servers that are not hardware NTP appliances, but rather general purpose computers running a software NTP implementation, like \texttt{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 \texttt{xleave} option to the server directive in \texttt{/etc/chrony.conf}:

\begin{verbatim}
server ntp.local minpoll 0 maxpoll 0 xleave
\end{verbatim}
29.5. 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
```

29.6. 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 29.3. 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 29.4. Reporting the transmit, receive timestamping and interleaved mode for each NTP source

```
# chronyc ntpdata
```

```
Remote address  : 203.0.113.15 (CB00710F)
Remote port     : 123
Local address   : 203.0.113.74 (CB00714A)
Leap status     : Normal
Version         : 4
Mode            : Server
Stratum         : 1
Poll interval   : 0 (1 seconds)
Precision       : -24 (0.000000060 seconds)
Root delay      : 0.000015 seconds
Root dispersion : 0.000015 seconds
Reference ID    : 47505300 (GPS)
Reference time  : Wed May 03 13:47:45 2017
Offset          : -0.000000134 seconds
Peer delay      : 0.000005396 seconds
Peer dispersion : 0.000002329 seconds
Response time   : 0.000152073 seconds
Jitter asymmetry: +0.00
NTP tests       : 111 111 1111
Interleaved     : Yes
Authenticated   : No
```
Example 29.5. 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
```

29.7. 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 29.6. Configuring chronyd to provide the system time using the other interface
```

```
bindaddress 203.0.113.74
hwtimestamp eth1
local stratum 1
```
CHAPTER 30. OVERVIEW OF NETWORK TIME SECURITY (NTS) IN CHRONY

Network Time Security (NTS) is an authentication mechanism for Network Time Protocol (NTP), designed to scale substantial clients. It verifies that the packets received from the server machines are unaltered while moving to the client machine. Network Time Security (NTS) includes a Key Establishment (NTS-KE) protocol that automatically creates the encryption keys used between the server and its clients.

30.1. ENABLING NETWORK TIME SECURITY (NTS) IN THE CLIENT CONFIGURATION FILE

By default, Network Time Security (NTS) is not enabled. You can enable NTS in the `/etc/chrony.conf`. For that, perform the following steps:

Prerequisites

- Server with the NTS support

Procedure

In the client configuration file:

1. Specify the server with the `nts` option in addition to the recommended `iburst` option.

   For example:
   ```
   server time.example.com iburst nts
   server nts.netnod.se iburst nts
   server ptbtime1.ptb.de iburst nts
   ```

2. To avoid repeating the Network Time Security-Key Establishment (NTS-KE) session during system boot, add the following line to `chrony.conf`, if it is not present:

   ```
   ntsdumpdir /var/lib/chrony
   ```

3. Add the following line to `/etc/sysconfig/network` to disable synchronization with Network Time Protocol (NTP) servers provided by DHCP:

   ```
   PEERNTP=no
   ```

4. Save your changes.

5. Restart the `chronyd` service:

   ```
   systemctl restart chronyd
   ```

Verification

- Verify if the NTS keys were successfully established:

  ```
  # chronyc -N authdata
  ```
<table>
<thead>
<tr>
<th>Name/IP address</th>
<th>Mode</th>
<th>KeyID</th>
<th>Type</th>
<th>KLen</th>
<th>Last Atmp</th>
<th>NAK</th>
<th>Cook</th>
<th>CLen</th>
</tr>
</thead>
<tbody>
<tr>
<td>time.example.com</td>
<td>NTS</td>
<td>1</td>
<td>15</td>
<td>256</td>
<td>33m</td>
<td>0</td>
<td>0</td>
<td>8</td>
</tr>
<tr>
<td>nts.sth1.ntp.se</td>
<td>NTS</td>
<td>1</td>
<td>15</td>
<td>256</td>
<td>33m</td>
<td>0</td>
<td>0</td>
<td>8</td>
</tr>
<tr>
<td>nts.sth2.ntp.se</td>
<td>NTS</td>
<td>1</td>
<td>15</td>
<td>256</td>
<td>33m</td>
<td>0</td>
<td>0</td>
<td>8</td>
</tr>
</tbody>
</table>

The KeyID, Type, and KLen should have non-zero values. If the value is zero, check the system log for error messages from chronyd.

- Verify the client is making NTP measurements:

```
# chronyc -N sources
```

<table>
<thead>
<tr>
<th>MS Name/IP address</th>
<th>Stratum</th>
<th>Poll</th>
<th>Reach</th>
<th>LastRx</th>
<th>Last sample</th>
</tr>
</thead>
<tbody>
<tr>
<td>time.example.com</td>
<td>3</td>
<td>6</td>
<td>377</td>
<td>45</td>
<td>+/- 11ms</td>
</tr>
<tr>
<td>nts.sth1.ntp.se</td>
<td>1</td>
<td>6</td>
<td>377</td>
<td>44</td>
<td>+/- 23ms</td>
</tr>
<tr>
<td>nts.sth2.ntp.se</td>
<td>1</td>
<td>6</td>
<td>377</td>
<td>44</td>
<td>+/- 22ms</td>
</tr>
</tbody>
</table>

The Reach column should have a non-zero value; ideally 377. If the value rarely gets 377 or never gets to 377, it indicates that NTP requests or responses are getting lost in the network.

Additional resources

- chrony.conf(5) man page

### 30.2. ENABLING NETWORK TIME SECURITY (NTS) ON THE SERVER

If you run your own Network Time Protocol (NTP) server, you can enable the server Network Time Security (NTS) support to facilitate its clients to synchronize securely.

If the NTP server is a client of other servers, that is, it is not a Stratum 1 server, it should use NTS or symmetric key for its synchronization.

**Prerequisites**

- Server private key in PEM format
- Server certificate with required intermediate certificates in PEM format

**Procedure**

1. Specify the private key and the certificate file in chrony.conf

   For example:
   
   ```
   ntsserverkey /etc/pki/tls/private/foo.example.net.key
   ntsservercert /etc/pki/tls/certs/foo.example.net.crt
   ```

2. Ensure that both the key and certificate files are readable by the chrony system user, by setting the group ownership.

   For example:
   
   ```
   chown :chrony /etc/pki/tls/*foo.example.net.*
   ```
3. Ensure the `ntsdumpdir /var/lib/chrony` directive is present in the `chrony.conf`.

4. Restart the `chronyd` service:

   ```
   systemctl restart chronyd
   ```

   **IMPORTANT**
   
   If the server has a firewall, it needs to allow both the **UDP 123** and **TCP 4460** ports for NTP and Network Time Security-Key Establishment (NTS-KE).

**Verification**

- Perform a quick test from a client machine with the following command:

   ```
   $ chronyd -Q -t 3 'server
   ```

   ```
   foo.example.net iburst nts maxsamples 1'
   2021-09-15T13:45:26Z chronyd version 4.1 starting (+CMDMON +NTP +REFCLOCK +RTC +PRIVDROP +SCFILTER +SIGND +ASYNCDNS +NTS +SECHASH +IPV6 +DEBUG)
   2021-09-15T13:45:26Z Disabled control of system clock
   2021-09-15T13:45:28Z System clock wrong by 0.002205 seconds (ignored)
   2021-09-15T13:45:28Z chronyd exiting
   ```

   The **System clock wrong** message indicates the NTP server is accepting NTS-KE connections and responding with NTS-protected NTP messages.

- Verify the NTS-KE connections and authenticated NTP packets observed on the server:

   ```
   # chronyc serverstats
   ```

   ```
   NTP packets received : 7
   NTP packets dropped : 0
   Command packets received : 22
   Command packets dropped : 0
   Client log records dropped : 0
   NTS-KE connections accepted: 1
   NTS-KE connections dropped : 0
   Authenticated NTP packets: 7
   ```

   If the value of the **NTS-KE connections accepted** and **Authenticated NTP packets** field is a non-zero value, it means that at least one client was able to connect to the NTS-KE port and send an authenticated NTP request.
CHAPTER 31. 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.

31.1. 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 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.
NOTE

In RHEL 9, the Secure copy protocol (SCP) is replaced with the SSH File Transfer Protocol (SFTP) by default. This is because SCP has already caused security issues, for example CVE-2020-15778.

If SFTP is unavailable or incompatible in your scenario, you can use the -O option to force use of the original SCP/RCP protocol.

For additional information, see the OpenSSH SCP protocol deprecation in Red Hat Enterprise Linux 9 article.

Two versions of SSH currently exist: version 1, and the newer version 2. The OpenSSH suite in RHEL supports only SSH version 2. It has an enhanced key-exchange algorithm that is not vulnerable to exploits known in version 1.

OpenSSH, as one of core cryptographic subsystems of RHEL, uses system-wide crypto policies. This ensures that weak cipher suites and cryptographic algorithms are disabled in the default configuration. To modify the policy, the administrator must either use the update-crypto-policies command to adjust the settings or manually opt out of the system-wide crypto policies.

The OpenSSH suite uses two sets of configuration files: one for client programs (that is, ssh, scp, and sftp), and another 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

31.2. CONFIGURING AND STARTING AN OPENSSH 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:

```bash
# 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/UzlTTku8GtXSz0S1+1IPegSy31v3L/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 ssd_config(5) man pages.

31.3. 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), ssd_config(5), and setsebool(8) man pages.
31.4. 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+SqwBKRNaU72oZfaCl
   joesec@localhost.example.com
   The key's randomart image is:
   +---[ECDSA 256]---+
   |.oo..o=++        |
   |.. o .oo .       |
   |. .. o. o        |
   |....o.+...       |
   |0.00.0 +S .      |
   |.=.+.   .o       |
   |E.*+.  .  . .    |
   |.=..+ +..  o     |
   |  .  oo*+o.      |
   +----[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.

### 31.5. 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**

1. 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...KZMzcQZzx
   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
   ```

2. 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
   ```

3. 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] $

4. 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

31.6. 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`:

```bash
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:

```bash
# systemctl mask sshd-keygen@rsa.service
```

- To exclude particular key types for SSH connections, comment out the relevant lines in `/etc/ssh/sshd_config`, and reload the `sshd` service. For example, to allow only Ed25519 host keys:

```bash
# 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/sshd_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:

```bash
# semanage port -a -t ssh_port_t -p tcp port_number
```

Furthermore, update `firewalld` configuration:

```bash
# 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.

### Root login

- `PermitRootLogin` is set to `prohibit-password` by default. 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.
CAUTION

Enabling logging in as the root user is not a secure practice because the administrator cannot audit which users run which privileged commands. For using administrative commands, log in and use `sudo` instead.

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 XI forwarding. Most applications are not able to run with this extension enabled anyway.
  
  By default, the `ForwardX11Trusted` option in the `/etc/ssh/sshd_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 XI 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 `Security hardening` title for more information.

Additional resources
31.7. 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.example.com:75
joesec@remote1.example.com:220
```

Additional resources

- ssh_config(5) and ssh(1) man pages.

31.8. 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.

### 31.9. 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 32. 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.

32.1. 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.

32.2. INSTALLING RSYSLOG DOCUMENTATION

The Rsyslog application has extensive online documentation that is available at https://www.rsyslog.com/doc/, but you can also install the rsyslog-doc documentation package locally.

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:

  # dnf install rsyslog-doc

Verification
32.3. 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.
- The policycoreutils-python-utils package is installed for the optional step using the semanage command.
- The firewalld service is running.

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:

   ```
   # firewall-cmd --zone=<zone-name> --permanent --add-port=30514/tcp
   success
   # firewall-cmd --reload
   ```

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/")
   }
   ```
### CHAPTER 32. CONFIGURING A REMOTE LOGGING SOLUTION

```ruby
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 TCP syslog reception
module(load="imtcp")

# Adding this ruleset to process 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="imtcp" port="30514" ruleset="remote1")
```

4. Save the changes to the `/etc/rsyslog.d/remotelog.conf` file.

5. Test the syntax of the `/etc/rsyslog.conf` file:

   ```bash
   # rsyslogd -N 1
   rsyslogd: version 8.1911.0-2.el8, config validation run (level 1), master config
   /etc/rsyslog.conf
   ```

6. Make sure the `rsyslog` service is running and enabled on the logging server:

   ```bash
   # systemctl status rsyslog
   ```

7. Restart the `rsyslog` service.

   ```bash
   # systemctl restart rsyslog
   ```

8. Optional: If `rsyslog` is not enabled, ensure the `rsyslog` service starts automatically after reboot:

   ```bash
   # systemctl enable rsyslog
   ```

Your log server is now configured to receive and store log files from the other systems in your environment.

**Additional resources**
- `rsyslogd(8)`, `rsyslog.conf(5)`, `semanage(8)`, and `firewall-cmd(1)` man pages.
32.4. 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.
- The system contains the policycoreutils-python-utils package, which provides the semanage command for adding a non-standard port to the SELinux configuration.

Procedure

1. Create a new file in the /etc/rsyslog.d/ directory named, for example, remotelog.conf, and insert the following content:

```conf
*.:* 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:

```bash
# logger test
```

2. On the server system, view the `/var/log/messages` log, for example:

```bash
# cat /var/log/remote/msg/hostame/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**

- `rsyslogd(8)` and `rsyslog.conf(5)` man pages.
- Documentation installed with the `rsyslog-doc` package at `file:///usr/share/doc/rsyslog/html/index.html`.

**32.5. CONFIGURING TLS-ENCRYPTED REMOTE LOGGING**

By default, Rsyslog sends remote-logging communication in the plain text format. If your scenario requires to secure this communication channel, you can encrypt it using TLS.

To use encrypted transport through TLS, configure both the server and the client. The server collects and analyzes the logs sent by one or more client systems.

You can use either the `ossl` network stream driver (OpenSSL) or the `gtls` stream driver (GnuTLS).

**NOTE**

If you have a separate system with higher security, for example, a system that is not connected to any network or has stricter authorizations, use the separate system as the certifying authority (CA).

**Prerequisites**

- You have `root` access to both the client and server systems.
- The `rsyslog` and `rsyslog openssl` packages are installed on the server and the client systems.
- If you use the `gtls` network stream driver, install the `rsyslog-gnutls` package instead of `rsyslog openssl`.
- If you generate certificates using the `certtool` command, install the `gnutls-utils` package.
- On your logging server, the following certificates are in the `/etc/pki/ca-trust/source/anchors/` directory and your system configuration is updated by using the `update-ca-trust` command:
- `ca-cert.pem` - a CA certificate that can verify keys and certificates on logging servers and clients.
- `server-cert.pem` - a public key of the logging server.
- `server-key.pem` - a private key of the logging server.

On your logging clients, the following certificates are in the `/etc/pki/ca-trust/source/anchors/` directory and your system configuration is updated by using `update-ca-trust`:

- `ca-cert.pem` - a CA certificate that can verify keys and certificates on logging servers and clients.
- `client-cert.pem` - a public key of a client.
- `client-key.pem` - a private key of a client.

## Procedure

1. Configure the server for receiving encrypted logs from your client systems:
   a. Create a new file in the `/etc/rsyslog.d/` directory named, for example, `securelogser.conf`.
   b. To encrypt the communication, the configuration file must contain paths to certificate files on your server, a selected authentication method, and a stream driver that supports TLS encryption. Add the following lines to the `/etc/rsyslog.d/securelogser.conf` file:

   ```
   # Set certificate files
   global(
     DefaultNetstreamDriverCAFile="/etc/pki/ca-trust/source/anchors/ca-cert.pem"
     DefaultNetstreamDriverCertFile="/etc/pki/ca-trust/source/anchors/server-cert.pem"
     DefaultNetstreamDriverKeyFile="/etc/pki/ca-trust/source/anchors/server-key.pem"
   )

   # TCP listener
   module(
     load="imtcp"
     PermittedPeer=["client1.example.com", "client2.example.com"]
     StreamDriver.AuthMode="x509/name"
     StreamDriver.Mode="1"
     StreamDriver.Name="ossl"
   )

   # Start up listener at port 514
   input(
     type="imtcp"
     port="514"
   )
   ```

   **NOTE**

   If you prefer the GnuTLS driver, use the `StreamDriver.Name="gtls"` configuration option. See the documentation installed with the `rsyslog-doc` package for more information about less strict authentication modes than `x509/name`.  

168
c. Save the changes to the `/etc/rsyslog.d/securelogser.conf` file.

d. Verify the syntax of the `/etc/rsyslog.conf` file and any 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
```

e. Make sure the `rsyslog` service is running and enabled on the logging server:

```
# systemctl status rsyslog
```

f. Restart the `rsyslog` service:

```
# systemctl restart rsyslog
```

g. Optional: If Rsyslog is not enabled, ensure the `rsyslog` service starts automatically after reboot:

```
# systemctl enable rsyslog
```

2. Configure clients for sending encrypted logs to the server:

a. On a client system, create a new file in the `/etc/rsyslog.d/` directory named, for example, `securelogcli.conf`.

b. Add the following lines to the `/etc/rsyslog.d/securelogcli.conf` file:

```
# Set certificate files
global(
    DefaultNetstreamDriverCAFile="/etc/pki/ca-trust/source/anchors/ca-cert.pem"
    DefaultNetstreamDriverCertFile="/etc/pki/ca-trust/source/anchors/client-cert.pem"
    DefaultNetstreamDriverKeyFile="/etc/pki/ca-trust/source/anchors/client-key.pem"
)

# Set up the action for all messages
  *.* action(
      type="omfwd"
      StreamDriver="ossl"
      StreamDriverMode="1"
      StreamDriverPermittedPeers="server.example.com"
      StreamDriverAuthMode="x509/name"
      target="server.example.com" port="514" protocol="tcp"
  )
```

```
NOTE
If you prefer the GnuTLS driver, use the StreamDriver.Name="gtls"
configuration option.
```

c. Save the changes to the `/etc/rsyslog.d/securelogser.conf` file.
d. Verify the syntax of the `/etc/rsyslog.conf` file and other files in the `/etc/rsyslog.d/` directory:

```bash
# rsyslogd -N 1
rsyslogd: version 8.191.0-2.el8, config validation run (level 1), master config
/etc/rsyslog.conf
```

e. Make sure the `rsyslog` service is running and enabled on the logging server:

```bash
# systemctl status rsyslog
```

f. Restart the `rsyslog` service:

```bash
# systemctl restart rsyslog
```

g. Optional: If Rsyslog is not enabled, ensure the `rsyslog` service starts automatically after reboot:

```bash
# 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:

   ```bash
   # logger test
   ```

2. On the server system, view the `/var/log/messages` log, for example:

   ```bash
   # 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 `certtool(1)`, `openssl(1)`, `update-ca-trust(8)`, `rsyslogd(8)`, and `rsyslog.conf(5)` man pages.
- Documentation installed with the `rsyslog-doc` package at `file:///usr/share/doc/rsyslog/html/index.html`.

### 32.6. 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**

- Rsyslog is installed on the server system.
- You are logged in as **root** on the server.
- The **policycoreutils-python-utils** package is installed for the optional step using the **semanage** command.
- The **firewalld** service is running.

**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
      # firewall-cmd --reload
      ```
   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. 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
```

4. Restart the **rsyslog** service.

```
# systemctl restart rsyslog
```

5. Optional: If **rsyslog** is not enabled, ensure the **rsyslog** service starts automatically after reboot:

```
# systemctl enable rsyslog
```

Additional resources

- **rsyslogd**(8), **rsyslog.conf**(5), **semanage**(8), and **firewall-cmd**(1) man pages.
- Documentation installed with the **rsyslog-doc** package at

### 32.7. 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:
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
32.8. LOAD BALANCING HELPER IN RSYSLOG

The **RebindInterval** setting specifies an interval at which the current connection is broken and is re-established. This setting applies to TCP, UDP, and RELP traffic. The load balancers perceive it as a new connection and forward the messages to another physical target system.

The **RebindInterval** setting proves to be helpful in scenarios when a target system has changed its IP address. The Rsyslog application caches the IP address when the connection establishes, therefore, the messages are sent to the same server. If the IP address changes, the UDP packets will be lost until the Rsyslog service restarts. Re-establishing the connection will ensure the IP to be resolved by DNS again.

```bash
action(type="omfwd" protocol="tcp" RebindInterval="250" target="example.com" port="514" …)
action(type="omfwd" protocol="udp" RebindInterval="250" target="example.com" port="514" …)
action(type="omrelp" RebindInterval="250" target="example.com" port="6514" …)
```

32.9. 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, **replclient.conf**, and insert the following content:
      ```bash
      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/replclient.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, `repserv.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/repserv.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**

- `rsyslogd(8)` and `rsyslog.conf(5)` man pages.
- Documentation installed with the `rsyslog-doc` package at file:///usr/share/doc/rsyslog/html/index.html.

### 32.10. 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:


### 32.11. 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 33. 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.

33.1. 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.

33.2. 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`

### 33.3. APPLYING A LOCAL LOGGING SYSTEM ROLE

Follow these steps to prepare and apply an Ansible 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 Core configures other systems.

On the control node:

- The ansible-core and rhel-system-roles packages are installed.

**IMPORTANT**

RHEL 8.0–8.5 provided access to a separate Ansible repository that contains Ansible Engine 2.9 for automation based on Ansible. Ansible Engine contains command-line utilities such as ansible, ansible-playbook, connectors such as docker and podman, and many plugins and modules. For information on how to obtain and install Ansible Engine, see the How to download and install Red Hat Ansible Engine Knowledgebase article.

RHEL 8.6 and 9.0 have introduced Ansible Core (provided as the ansible-core package), which contains the Ansible command-line utilities, commands, and a small set of built-in Ansible plugins. RHEL provides this package through the AppStream repository, and it has a limited scope of support. For more information, see the Scope of support for the Ansible Core package included in the RHEL 9 and RHEL 8.6 and later AppStream repositories Knowledgebase article.

- 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:
    - rhel-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:

```bash
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:

```bash
rsyslogd -N 1
rsyslogd: version 8.1911.0-6.el8, config validation run (level 1), master config /etc/rsyslog.conf
```

2. Verify that the system sends messages to the log:

   a. Send a test message:

```bash
logger test
```

   b. View the `/var/log/messages` log, for example:

```bash
# 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`.

### 33.4. 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 Core configures other systems.

  On the control node:

  - Red Hat Ansible Core 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:

```yaml
---
- name: Deploying files input and configured files output
  hosts: all
  roles:
    - linux-system-roles.logging
  vars:
    logging_inputs:
      - name: files_input
        type: basics
    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_input]
      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.

2. Optional: Verify playbook syntax.

   ```bash
   # ansible-playbook --syntax-check playbook.yml
   ```

3. Run the playbook on your inventory file:

   ```bash
   # ansible-playbook -i inventory_file /path/to/file/playbook.yml
   ```

Verification

1. Test the syntax of the `/etc/rsyslog.conf` file:
2. Verify that the system sends messages that contain the `error` string to the log:
   a. Send a test message:
      ```
      # logger error
      ```
   b. 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`

33.5. APPLYING A REMOTE LOGGING SOLUTION USING THE LOGGING SYSTEM ROLE

Follow these steps to prepare and apply a Red Hat Ansible Core 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 Core configures other systems.
  On the control node:
    - The `ansible-core` and `rhel-system-roles` packages are 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:

```yaml
# vi logging-playbook.yml
```

b. Insert the following content into the file:

```yaml
---
- name: Deploying remote input and remote_files output
  hosts: server
  roles:
    - rhel-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:
    - rhel-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
   ```
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

### 33.6. 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 RHEL System Role to configure secure transfer of logs using Red Hat Ansible Automation Platform.

#### 33.6.1. 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
```
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
```

### 33.6.2. 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:

```yaml
---
- 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:
```
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
```

### 33.7. 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.

### 33.7.1. 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:

```yaml
- 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 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_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.

   ```bash
   # ansible-playbook --syntax-check playbook.yml
   ```

3. Run the playbook:

   ```bash
   # ansible-playbook -i inventory_file playbook.yml
   ```
33.7.2. 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:

```yaml
---
- 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 
ode 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.

```bash
# ansible-playbook --syntax-check playbook.yml
```

3. Run the playbook:

```bash
# ansible-playbook -i inventory_file playbook.yml
```

### 33.8. 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.