



Administration Guide

SUSE Linux Enterprise Server 12 SP2



Administration Guide

SUSE Linux Enterprise Server 12 SP2

Covers system administration tasks like maintaining, monitoring and customizing an initially installed system.

Publication Date: October 19, 2016

SUSE LLC
10 Canal Park Drive
Suite 200
Cambridge MA 02141
USA

<https://www.suse.com/documentation> 

Copyright © 2006– 2016 SUSE LLC and contributors. All rights reserved.

Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.2 or (at your option) version 1.3; with the Invariant Section being this copyright notice and license. A copy of the license version 1.2 is included in the section entitled “GNU Free Documentation License”.

For SUSE trademarks, see <http://www.suse.com/company/legal/> . All other third-party trademarks are the property of their respective owners. Trademark symbols (®, ™ etc.) denote trademarks of SUSE and its affiliates. Asterisks (*) denote third-party trademarks.

All information found in this book has been compiled with utmost attention to detail. However, this does not guarantee complete accuracy. Neither SUSE LLC, its affiliates, the authors nor the translators shall be held liable for possible errors or the consequences thereof.

Contents

About This Guide **xx**

I COMMON TASKS **1**

1 Bash and Bash Scripts **2**

1.1 What is “The Shell”? **2**

Knowing the Bash Configuration Files **2** • The Directory Structure **3**

1.2 Writing Shell Scripts **8**

1.3 Redirecting Command Events **9**

1.4 Using Aliases **10**

1.5 Using Variables in Bash **10**

Using Argument Variables **11** • Using Variable Substitution **12**

1.6 Grouping and Combining Commands **13**

1.7 Working with Common Flow Constructs **14**

The if Control Command **14** • Creating Loops With the **for** Command **15**

1.8 For More Information **15**

2 sudo **16**

2.1 Basic **sudo** Usage **16**

Running a Single Command **16** • Starting a Shell **17** • Environment Variables **18**

2.2 Configuring **sudo** **18**

Editing the Configuration Files **19** • Basic sudoers Configuration Syntax **19** • Rules in sudoers **21**

2.3	Common Use Cases	23
	Using sudo without root Password	23 • Using sudo with X.Org Applications
2.4	More Information	25
3	YaST Online Update	26
3.1	The Online Update Dialog	27
3.2	Installing Patches	28
3.3	Automatic Online Update	29
4	YaST in Text Mode	31
4.1	Navigation in Modules	32
4.2	Restriction of Key Combinations	34
4.3	YaST Command Line Options	34
	Starting the Individual Modules	34 • Installing Packages from the Command Line
	Command Line Parameters of the YaST Modules	35
5	Managing Software with Command Line Tools	36
5.1	Using Zypper	36
	General Usage	36 • Installing and Removing Software with Zypper
	Updating Software with Zypper	42 • Identifying Processes and Services Using Deleted Files
	Managing Repositories with Zypper	48 • Querying Repositories and Packages with Zypper
	Configuring Zypper	51 • Troubleshooting
	Zypper Rollback Feature on Btrfs File System	51 • For More Information
5.2	RPM—the Package Manager	52
	Verifying Package Authenticity	53 • Managing Packages: Install, Update, and Uninstall
	Delta RPM Packages	54 • RPM Queries
	Installing and Compiling Source Packages	58 • Compiling RPM Packages with build
	Tools for RPM Archives and the RPM Database	60

6 System Recovery and Snapshot Management with Snapper 62

6.1 Default Setup 62

Types of Snapshots 64 • Directories That Are Excluded from Snapshots 64 • Customizing the Setup 66

6.2 Using Snapper to Undo Changes 69

Undoing YaST and Zypper Changes 70 • Using Snapper to Restore Files 75

6.3 System Rollback by Booting from Snapshots 77

Accessing and Identifying Snapshot Boot Entries 79 • Limitations 80

6.4 Creating and Modifying Snapper Configurations 81

Managing Existing Configurations 82

6.5 Manually Creating and Managing Snapshots 86

Snapshot Metadata 86 • Creating Snapshots 88 • Modifying Snapshot Metadata 89 • Deleting Snapshots 89

6.6 Automatic Snapshot Clean-Up 91

Cleaning Up Numbered Snapshots 91 • Cleaning Up Timeline Screenshots 93 • Cleaning Up Snapshot Pairs That Do Not Differ 95 • Cleaning Up Manually Created Snapshots 95 • Adding Disk Quota Support 95

6.7 Frequently Asked Questions 97

7 Remote Access with VNC 99

7.1 The **vncviewer** Client 99

Connecting Using the vncviewer CLI 99 • Connecting Using the vncviewer GUI 99 • Notification of Unencrypted Connections 100

7.2 One-time VNC Sessions 100

Available Configurations 101 • Initiating a One-time VNC Session 101 • Configuring One-time VNC Sessions 101

7.3 Persistent VNC Sessions 102

Connecting to a Persistent VNC Session 104 • Configuring Persistent VNC Sessions 104

8 File Synchronization 105

- 8.1 Available Data Synchronization Software 105
 - CVS 106 • rsync 106
- 8.2 Determining Factors for Selecting a Program 106
 - Client/Server versus Peer-to-Peer 106 • Portability 107 • Interactive versus Automatic 107 • Conflicts: Incidence and Solution 107 • Selecting and Adding Files 107 • History 107 • Data Volume and Hard Disk Requirements 108 • GUI 108 • User Friendliness 108 • Security against Attacks 108 • Protection against Data Loss 108
- 8.3 Introduction to CVS 109
 - Configuring a CVS Server 110 • Using CVS 110
- 8.4 Introduction to rsync 112
 - Configuration and Operation 112
- 8.5 For More Information 113

II SYSTEM 115

9 32-Bit and 64-Bit Applications in a 64-Bit System Environment 116

- 9.1 Runtime Support 116
- 9.2 Software Development 117
- 9.3 Software Compilation on Biarch Platforms 118
- 9.4 Kernel Specifications 119

10 Booting a Linux System 121

- 10.1 The Linux Boot Process 121
- 10.2 initramfs 123
- 10.3 Init on initramfs 124

11 The Boot Loader GRUB 2 127

- 11.1 Main Differences between GRUB Legacy and GRUB 2 127
- 11.2 Configuration File Structure 127
 - The File `/boot/grub2/grub.cfg` 129 • The File `/etc/default/grub` 129 • Scripts in `/etc/grub.d` 132 • Mapping between BIOS Drives and Linux Devices 133 • Editing Menu Entries during the Boot Procedure 133 • Setting a Boot Password 135
- 11.3 Configuring the Boot Loader with YaST 136
 - Modifying the Boot Loader Location 137 • Adjusting the Disk Order 137 • Configuring Advanced Options 138
- 11.4 Differences in Terminal Usage on z Systems 141
 - Limitations 141 • Key Combinations 141
- 11.5 Helpful GRUB 2 Commands 143
- 11.6 More Information 145

12 UEFI (Unified Extensible Firmware Interface) 146

- 12.1 Secure Boot 146
 - Implementation on SUSE Linux Enterprise 147 • MOK (Machine Owner Key) 151 • Booting a Custom Kernel 151 • Using Non-Inbox Drivers 153 • Features and Limitations 154
- 12.2 For More Information 155

13 The systemd Daemon 156

- 13.1 The systemd Concept 156
 - What Is systemd 156 • Unit File 157
- 13.2 Basic Usage 158
 - Managing Services in a Running System 158 • Permanently Enabling/Disabling Services 160
- 13.3 System Start and Target Management 161
 - Targets Compared to Runlevels 162 • Debugging System Start-Up 165 • System V Compatibility 168

13.4	Managing Services with YaST	169
13.5	Customization of <code>systemd</code>	170
	Customizing Service Files	170 • Creating “Drop-in” Files 171 • Creating Custom Targets 171
13.6	Advanced Usage	172
	Cleaning Temporary Directories	172 • System Log 173 • Snapshots 173 • Loading Kernel Modules 174 • Performing Actions Before Loading a Service 174 • Kernel Control Groups (cgroups) 175 • Terminating Services (Sending Signals) 176 • Debugging Services 177
13.7	More Information	178
14	<code>journalctl</code>: Query the <code>systemd</code> Journal	179
14.1	Making the Journal Persistent	179
14.2	<code>journalctl</code> Useful Switches	180
14.3	Filtering the Journal Output	181
	Filtering Based on a Boot Number	181 • Filtering Based on Time Interval 182 • Filtering Based on Fields 182
14.4	Investigating <code>systemd</code> Errors	183
14.5	Journal Configuration	184
	Changing the Journal Size Limit	185 • Forwarding the Journal to <code>/dev/ttyX</code> 185 • Forwarding the Journal to Syslog Facility 185
14.6	Using YaST to Filter the <code>systemd</code> Journal	186
15	Basic Networking	187
15.1	IP Addresses and Routing	190
	IP Addresses	190 • Netmasks and Routing 190
15.2	IPv6—The Next Generation Internet	192
	Advantages	193 • Address Types and Structure 194 • Coexistence of IPv4 and IPv6 199 • Configuring IPv6 200 • For More Information 200
15.3	Name Resolution	201

- 15.4 Configuring a Network Connection with YaST **202**
Configuring the Network Card with YaST **203** • IBM z Systems: Configuring Network Devices **214**
- 15.5 Configuring a Network Connection Manually **216**
The **wicked** Network Configuration **216** • Configuration Files **224** • Testing the Configuration **235** • Unit Files and Start-Up Scripts **238**
- 15.6 Basic Router Setup **239**
- 15.7 Setting Up Bonding Devices **241**
Hotplugging of Bonding Slaves **243**
- 15.8 Setting Up Team Devices for Network Teaming **244**
Use Case: Loadbalancing with Network Teaming **246** • Use Case: Failover with Network Teaming **247**
- 15.9 Software-Defined Networking with Open vSwitch **249**
Advantages of Open vSwitch **249** • Installing Open vSwitch **249** • Overview of Open vSwitch Daemons and Utilities **250** • Creating a Bridge with Open vSwitch **251** • Using Open vSwitch Directly with KVM **252** • Using Open vSwitch with libvirt **254** • For More Information **255**
- 16 Printer Operation 256**
- 16.1 The CUPS Workflow **257**
- 16.2 Methods and Protocols for Connecting Printers **258**
- 16.3 Installing the Software **258**
- 16.4 Network Printers **259**
- 16.5 Configuring CUPS with Command Line Tools **260**
- 16.6 Printing from the Command Line **262**
- 16.7 Special Features in SUSE Linux Enterprise Server **262**
CUPS and Firewall **262** • Browsing for Network Printers **263** • PPD Files in Various Packages **263**

- 16.8 Troubleshooting 264
Printers without Standard Printer Language Support 264 • No Suitable PPD File Available for a PostScript Printer 265 • Network Printer Connections 265 • Defective Printouts without Error Message 268 • Disabled Queues 268 • CUPS Browsing: Deleting Print Jobs 268 • Defective Print Jobs and Data Transfer Errors 269 • Debugging CUPS 269 • For More Information 270

17 The X Window System 271

- 17.1 Installing and Configuring Fonts 271
Showing Installed Fonts 273 • Viewing Fonts 273 • Querying Fonts 274 • Installing Fonts 274 • Configuring the Appearance of Fonts 275
- 17.2 For More Information 284

18 Accessing File Systems with FUSE 285

- 18.1 Configuring FUSE 285
- 18.2 Mounting an NTFS Partition 285
- 18.3 For More Information 286

19 Dynamic Kernel Device Management with udev 287

- 19.1 The /dev Directory 287
- 19.2 Kernel uevents and udev 287
- 19.3 Drivers, Kernel Modules and Devices 288
- 19.4 Booting and Initial Device Setup 289
- 19.5 Monitoring the Running udev Daemon 289
- 19.6 Influencing Kernel Device Event Handling with udev Rules 290
Using Operators in udev Rules 292 • Using Substitutions in udev Rules 293 • Using udev Match Keys 294 • Using udev Assign Keys 295

19.7	Persistent Device Naming	297
19.8	Files used by udev	298
19.9	For More Information	298
20	Live Patching the Linux Kernel Using kGraft	299
20.1	Advantages of kGraft	299
20.2	Low-level Function of kGraft	300
20.3	Installing kGraft Patches	301
	Activation of SLE Live Patching	301 • Updating System 301
20.4	Removing a kGraft Patch	302
20.5	Stuck Kernel Execution Threads	302
20.6	The kgr Tool	303
20.7	Scope of kGraft Technology	303
20.8	Scope of SLE Live Patching	303
20.9	Interaction with the Support Processes	304
21	Special System Features	305
21.1	Information about Special Software Packages	305
	The bash Package and /etc/profile	305 • The cron Package 306 • Stop-
	ping Cron Status Messages	307 • Log Files: Package logrotate 307 • The
	locate Command	308 • The ulimit Command 309 • The free Com-
	mand	310 • Man Pages and Info Pages 310 • Selecting Man Pages Using
	the man Command	310 • Settings for GNU Emacs 311
21.2	Virtual Consoles	312
21.3	Keyboard Mapping	312
21.4	Language and Country-Specific Settings	313
	Some Examples	314 • Locale Settings in ~/.i18n 315 • Settings for Lan-
	guage Support	315 • For More Information 316

III SERVICES 317

22 Time Synchronization with NTP 318

- 22.1 Configuring an NTP Client with YaST 318
 - Basic Configuration 318 • Changing Basic Configuration 319
- 22.2 Manually Configuring NTP in the Network 322
- 22.3 Dynamic Time Synchronization at Runtime 322
- 22.4 Setting Up a Local Reference Clock 323
- 22.5 Clock Synchronization to an External Time Reference (ETR) 324

23 The Domain Name System 325

- 23.1 DNS Terminology 325
- 23.2 Installation 326
- 23.3 Configuration with YaST 326
 - Wizard Configuration 326 • Expert Configuration 329
- 23.4 Starting the BIND Name Server 337
- 23.5 The /etc/named.conf Configuration File 339
 - Important Configuration Options 340 • Logging 341 • Zone Entries 342
- 23.6 Zone Files 343
- 23.7 Dynamic Update of Zone Data 347
- 23.8 Secure Transactions 348
- 23.9 DNS Security 349
- 23.10 For More Information 349

24 DHCP 350

- 24.1 Configuring a DHCP Server with YaST 351
 - Initial Configuration (Wizard) 351 • DHCP Server Configuration (Expert) 356
- 24.2 DHCP Software Packages 361

- 24.3 The DHCP Server `dhcpd` 362
 - Clients with Fixed IP Addresses 364 • The SUSE Linux Enterprise Server Version 365
- 24.4 For More Information 365
- 25 Sharing File Systems with NFS 366**
 - 25.1 Terminology 366
 - 25.2 Installing NFS Server 367
 - 25.3 Configuring NFS Server 367
 - Exporting File Systems with YaST 367 • Exporting File Systems Manually 369 • NFS with Kerberos 371
 - 25.4 Configuring Clients 372
 - Importing File Systems with YaST 372 • Importing File Systems Manually 373 • Parallel NFS (pNFS) 375
 - 25.5 For More Information 376
- 26 Samba 377**
 - 26.1 Terminology 377
 - 26.2 Installing a Samba Server 378
 - 26.3 Starting and Stopping Samba 379
 - 26.4 Configuring a Samba Server 379
 - Configuring a Samba Server with YaST 379 • Configuring the Server Manually 382
 - 26.5 Configuring Clients 386
 - Configuring a Samba Client with YaST 386
 - 26.6 Samba as Login Server 386
 - 26.7 Samba Server in the Network with Active Directory 387
 - 26.8 Advanced Topics 389
 - Transparent File Compression on Btrfs 389 • Snapshots 390

- 26.9 For More Information 398
- 27 On-Demand Mounting with Autofs 399**
 - 27.1 Installation 399
 - 27.2 Configuration 399
 - The Master Map File 399 • Map Files 401
 - 27.3 Operation and Debugging 402
 - Controlling the autofs Service 402 • Debugging the Automounter Problems 403
 - 27.4 Auto-Mounting an NFS Share 404
 - 27.5 Advanced Topics 405
 - /net Mount Point 405 • Using Wild Cards to Auto-Mount Subdirectories 406 • Auto-Mounting CIFS File System 406
- 28 SLP 407**
 - 28.1 The SLP Front-End **slptool** 407
 - 28.2 Providing Services via SLP 408
 - Setting up an SLP Installation Server 410
 - 28.3 For More Information 410
- 29 The Apache HTTP Server 411**
 - 29.1 Quick Start 411
 - Requirements 411 • Installation 412 • Start 412
 - 29.2 Configuring Apache 413
 - Apache Configuration Files 413 • Configuring Apache Manually 416 • Configuring Apache with YaST 421
 - 29.3 Starting and Stopping Apache 427
 - 29.4 Installing, Activating, and Configuring Modules 429
 - Module Installation 430 • Activation and Deactivation 430 • Base and Extension Modules 430 • Multiprocessing Modules 433 • External Modules 435 • Compilation 436

- 29.5 Enabling CGI Scripts 436
 - Apache Configuration 437 • Running an Example Script 438 • CGI Troubleshooting 438
- 29.6 Setting Up a Secure Web Server with SSL 439
 - Creating an SSL Certificate 439 • Configuring Apache with SSL 443
- 29.7 Running Multiple Apache Instances on the Same Server 445
- 29.8 Avoiding Security Problems 448
 - Up-to-Date Software 449 • DocumentRoot Permissions 449 • File System Access 449 • CGI Scripts 449 • User Directories 450
- 29.9 Troubleshooting 450
- 29.10 For More Information 451
 - Apache 2.4 451 • Apache Modules 452 • Development 452 • Miscellaneous Sources 453
- 30 Setting Up an FTP Server with YaST 454**
 - 30.1 Starting the FTP Server 454
 - 30.2 FTP General Settings 455
 - 30.3 FTP Performance Settings 456
 - 30.4 Authentication 457
 - 30.5 Expert Settings 457
 - 30.6 For More Information 457
- 31 The Proxy Server Squid 458**
 - 31.1 Some Facts about Proxy Caches 458
 - Squid and Security 459 • Multiple Caches 459 • Caching Internet Objects 460
 - 31.2 System Requirements 460
 - RAM 461 • CPU 461 • Size of the Disk Cache 461 • Hard Disk/SSD Architecture 462

- 31.3 Basic Usage of Squid 462
 - Starting Squid 462 • Checking Whether Squid Is Working 463 • Stopping, Reloading, and Restarting Squid 465 • Removing Squid 465 • Local DNS Server 465
- 31.4 The /etc/squid/squid.conf Configuration File 466
 - General Configuration Options 467 • Options for Access Controls 470
- 31.5 Configuring a Transparent Proxy 472
- 31.6 Using the Squid Cache Manager CGI Interface (cachemgr.cgi) 475
- 31.7 squidGuard 477
- 31.8 Cache Report Generation with Calamaris 479
- 31.9 For More Information 479
- 32 Web Based Enterprise Management Using SFCB 481**
 - 32.1 Introduction and Basic Concept 481
 - 32.2 Setting Up SFCB 483
 - Installing Additional Providers 484 • Starting, Stopping and Checking Status for SFCB 485 • Ensuring Secure Access 486
 - 32.3 SFCB CIMOM Configuration 488
 - Environment Variables 488 • Command Line Options 489 • SFCB Configuration File 491
 - 32.4 Advanced SFCB Tasks 503
 - Installing CMPI Providers 503 • Testing SFCB 507 • Command Line CIM Client: wbemcli 509
 - 32.5 For More Information 511

IV MOBILE COMPUTERS 513

33 Mobile Computing with Linux 514

33.1 Laptops 514

Power Conservation 514 • Integration in Changing Operating Environments 515 • Software Options 517 • Data Security 522

33.2 Mobile Hardware 523

33.3 Cellular Phones and PDAs 524

33.4 For More Information 524

34 Using NetworkManager 525

34.1 Use Cases for NetworkManager 525

34.2 Enabling or Disabling NetworkManager 525

34.3 Configuring Network Connections 526

Managing Wired Network Connections 528 • Managing Wireless Network Connections 528 • Configuring Your Wi-Fi/Bluetooth Card as an Access Point 529 • NetworkManager and VPN 529

34.4 NetworkManager and Security 531

User and System Connections 531 • Storing Passwords and Credentials 531

34.5 Frequently Asked Questions 532

34.6 Troubleshooting 533

34.7 For More Information 534

35 Power Management 535

35.1 Power Saving Functions 535

35.2 Advanced Configuration and Power Interface (ACPI) 536

Controlling the CPU Performance 537 • Troubleshooting 537

35.3 Rest for the Hard Disk 539

35.4 Troubleshooting 540

CPU Frequency Does Not Work 540

35.5 For More Information 541

V TROUBLESHOOTING 542

36 Help and Documentation 543

36.1 Documentation Directory 543

SUSE Manuals 544 • Package Documentation 544

36.2 Man Pages 545

36.3 Info Pages 546

36.4 Online Resources 547

37 Gathering System Information for Support 549

37.1 Displaying Current System Information 549

37.2 Collecting System Information with Supportconfig 550

Creating a Service Request Number 550 • Upload Targets 550 • Creating a Supportconfig Archive with YaST 551 • Creating a Supportconfig Archive from Command Line 553 • Common Supportconfig Options 553

37.3 Submitting Information to Global Technical Support 554

37.4 Analyzing System Information 556

SCA Command Line Tool 557 • SCA Appliance 558 • Developing Custom Analysis Patterns 570

37.5 Gathering Information During the Installation 570

37.6 Support of Kernel Modules 571

Technical Background 571 • Working with Unsupported Modules 572

37.7 For More Information 573

38 Common Problems and Their Solutions 574

38.1 Finding and Gathering Information 574

38.2	Installation Problems 577
	Checking Media 577 • No Bootable DVD Drive Available 578 • Booting from Installation Media Fails 579 • Fails to Boot 580 • Fails to Launch Graphical Installer 582 • Only Minimalistic Boot Screen Started 584 • Log Files 584
38.3	Boot Problems 585
	Fails to Load the GRUB 2 Boot Loader 585 • No Graphical Login 585 • Root Btrfs Partition Cannot Be Mounted 586 • Force Checking Root Partitions 586
38.4	Login Problems 586
	Valid User Name and Password Combinations Fail 587 • Valid User Name and Password Not Accepted 588 • Login to Encrypted Home Partition Fails 590 • Login Successful but GNOME Desktop Fails 591
38.5	Network Problems 591
	NetworkManager Problems 595
38.6	Data Problems 596
	Managing Partition Images 596 • Using the Rescue System 597
38.7	IBM z Systems: Using initrd as a Rescue System 604
A	Documentation Updates 606
A.1	November 2016 (Initial Release of SUSE Linux Enterprise Server 12 SP2) 606
A.2	March 2016 (Maintenance Release of SUSE Linux Enterprise Server 12 SP1) 608
A.3	December 2015 (Initial Release of SUSE Linux Enterprise Server 12 SP1) 609
A.4	February 2015 (Documentation Maintenance Update) 612
A.5	October 2014 (Initial Release of SUSE Linux Enterprise Server 12) 613
B	An Example Network 619
C	GNU Licenses 620
C.1	GNU Free Documentation License 620

About This Guide

This guide is intended for use by professional network and system administrators during the operation of SUSE® Linux Enterprise. As such, it is solely concerned with ensuring that SUSE Linux Enterprise is properly configured and that the required services on the network are available to allow it to function properly as initially installed. This guide does not cover the process of ensuring that SUSE Linux Enterprise offers proper compatibility with your enterprise's application software or that its core functionality meets those requirements. It assumes that a full requirements audit has been done and the installation has been requested or that a test installation for such an audit has been requested.

This guide contains the following:

Support and Common Tasks

SUSE Linux Enterprise offers a wide range of tools to customize various aspects of the system. This part introduces a few of them. A breakdown of available device technologies, high availability configurations, and advanced administration possibilities introduces the system to the administrator.

System

Learn more about the underlying operating system by studying this part. SUSE Linux Enterprise supports several hardware architectures and you can use this to adapt your own applications to run on SUSE Linux Enterprise. The boot loader and boot procedure information assists you in understanding how your Linux system works and how your own custom scripts and applications may blend in with it.

Services

SUSE Linux Enterprise is designed to be a network operating system. It offers a wide range of network services, such as DNS, DHCP, Web, proxy, and authentication services, and integrates well into heterogeneous environments including MS Windows clients and servers.


Mobile Computers

Laptops, and the communication between mobile devices like PDAs, or cellular phones and SUSE Linux Enterprise need some special attention. Take care for power conservation and for the integration of different devices into a changing network environment. Also get in touch with the background technologies that provide the needed functionality.

Troubleshooting

Provides an overview of where to find help and additional documentation in case you need more information or want to perform specific tasks with your system. Also find a compilation of the most frequent problems and annoyances and learn how to solve these issues on your own.

Many chapters in this manual contain links to additional documentation resources. This includes additional documentation that is available on the system and documentation available on the Internet.

For an overview of the documentation available for your product and the latest documentation updates, refer to <http://www.suse.com/doc> .

1 Available Documentation

We provide HTML and PDF versions of our books in different languages. The following manuals for users and administrators are available for this product:

Article “Installation Quick Start”

Lists the system requirements and guides you step-by-step through the installation of SUSE Linux Enterprise Server from DVD, or from an ISO image.

Book “Deployment Guide”

Shows how to install single or multiple systems and how to exploit the product inherent capabilities for a deployment infrastructure. Choose from various approaches, ranging from a local installation or a network installation server to a mass deployment using a remote-controlled, highly-customized, and automated installation technique.

Administration Guide

Covers system administration tasks like maintaining, monitoring and customizing an initially installed system.

Book “Virtualization Guide”

Describes virtualization technology in general, and introduces libvirt—the unified interface to virtualization—and detailed information on specific hypervisors.

Book “Storage Administration Guide”

Provides information about how to manage storage devices on a SUSE Linux Enterprise Server.

Book “AutoYaST”

AutoYaST is a system for installing one or more SUSE Linux Enterprise Server systems automatically and without user intervention, using an AutoYaST profile that contains installation and configuration data. The manual guides you through the basic steps of auto-installation: preparation, installation, and configuration.

Book “Security Guide”

Introduces basic concepts of system security, covering both local and network security aspects. Shows how to use the product inherent security software like AppArmor or the auditing system that reliably collects information about any security-relevant events.

Book “Security and Hardening Guide”

Deals with the particulars of installing and setting up a secure SUSE Linux Enterprise Server, and additional post-installation processes required to further secure and harden that installation. Supports the administrator with security-related choices and decisions.

Book “System Analysis and Tuning Guide”


An administrator's guide for problem detection, resolution and optimization. Find how to inspect and optimize your system by means of monitoring tools and how to efficiently manage resources. Also contains an overview of common problems and solutions and of additional help and documentation resources.

Book “Subscription Management Tool for SLES 12 SP2”

An administrator's guide to Subscription Management Tool—a proxy system for SUSE Customer Center with repository and registration targets. Learn how to install and configure a local SMT server, how to mirror and manage repositories, how to manage client machines, and how to configure clients to use SMT.

Book “GNOME User Guide”

Introduces the GNOME desktop of SUSE Linux Enterprise Server. It guides you through using and configuring the desktop and helps you perform key tasks. It is intended mainly for end users who want to make efficient use of GNOME as their default desktop.

Find HTML versions of most product manuals in your installed system under `/usr/share/doc/manual`. The latest documentation updates are available at <http://www.suse.com/documentation/>  where you can download the documentation for your product in various formats.

2 Feedback

Several feedback channels are available:

Bugs and Enhancement Requests

For services and support options available for your product, refer to <http://www.suse.com/support/>.

To report bugs for a product component, go to <https://scc.suse.com/support/requests>, log in, and click *Create New*.

User Comments

We want to hear your comments about and suggestions for this manual and the other documentation included with this product. Use the User Comments feature at the bottom of each page in the online documentation or go to <http://www.suse.com/documentation/feedback.html> and enter your comments there.





Mail

For feedback on the documentation of this product, you can also send a mail to doc-team@suse.com. Make sure to include the document title, the product version and the publication date of the documentation. To report errors or suggest enhancements, provide a concise description of the problem and refer to the respective section number and page (or URL).

3 Documentation Conventions

The following notices and typographical conventions are used in this documentation:

- /etc/passwd: directory names and file names
- PLACEHOLDER: replace PLACEHOLDER with the actual value
- PATH: the environment variable PATH
- ls, --help: commands, options, and parameters
- user: users or groups
- package name: name of a package
- Alt, Alt-F1: a key to press or a key combination; keys are shown in uppercase as on a keyboard

- *File, File > Save As*: menu items, buttons
-  This paragraph is only relevant for the AMD64/Intel 64 architecture. The arrows mark the beginning and the end of the text block. 
-  This paragraph is only relevant for the architectures z Systems and POWER. The arrows mark the beginning and the end of the text block. 
- *Dancing Penguins* (Chapter *Penguins*, ↑Another Manual): This is a reference to a chapter in another manual.
- Commands that must be run with root privileges. Often you can also prefix these commands with the sudo command to run them.

```
root # command
```

- Commands that can be run by non-privileged users.

```
tux > command
```

- Notices



Warning: Warning Notice

Vital information you must be aware of before proceeding. Warns you about security issues, potential loss of data, damage to hardware, or physical hazards.



Important: Important Notice

Important information you should be aware of before proceeding.



Note: Note Notice





Additional information, for example about differences in software versions.



Tip: Tip Notice

Helpful information, like a guideline or a piece of practical advice.

4 About the Making of This Documentation

This documentation is written in SUSEDoc, a subset of DocBook 5 (<http://www.docbook.org>) . The XML source files were validated by [jing](https://code.google.com/p/jing-trang/) (see <https://code.google.com/p/jing-trang/> ) , processed by [xsltproc](#) , and converted into XSL-FO using a customized version of Norman Walsh's stylesheets. The final PDF is formatted through FOP from Apache Software Foundation (<https://xmlgraphics.apache.org/fop>) . The open source tools and the environment used to build this documentation are provided by the DocBook Authoring and Publishing Suite (DAPS). The project's home page can be found at <https://github.com/openSUSE/daps> .

The XML source code of this documentation can be found at <https://github.com/SUSE/doc-sle> .

I Common Tasks

- 1 Bash and Bash Scripts **2**
- 2 sudo **16**
- 3 YaST Online Update **26**
- 4 YaST in Text Mode **31**
- 5 Managing Software with Command Line Tools **36**
- 6 System Recovery and Snapshot Management with Snapper **62**
- 7 Remote Access with VNC **99**
- 8 File Synchronization **105**

1 Bash and Bash Scripts

These days many people use computers with a graphical user interface (GUI) like GNOME. Although they offer lots of features, their use is limited when it comes to the execution of automated tasks. Shells are a good addition to GUIs and this chapter gives you an overview of some aspects of shells, in this case Bash.

1.1 What is “The Shell”?

Traditionally, *the* shell is Bash (Bourne again Shell). When this chapter speaks about “the shell” it means Bash. There are actually more available shells than Bash (ash, csh, ksh, zsh, ...), each employing different features and characteristics. If you need further information about other shells, search for *shell* in YaST.

1.1.1 Knowing the Bash Configuration Files

A shell can be invoked as an:

1. **Interactive login shell.** This is used when logging in to a machine, invoking Bash with the `--login` option or when logging in to a remote machine with SSH.
2. **“Ordinary” interactive shell.** This is normally the case when starting xterm, konsole, gnome-terminal or similar tools.
3. **Non-interactive shell.** This is used when invoking a shell script at the command line.

Depending on which type of shell you use, different configuration files are being read. The following tables show the login and non-login shell configuration files.

TABLE 1.1: BASH CONFIGURATION FILES FOR LOGIN SHELLS

File	Description
<u>/etc/profile</u>	Do not modify this file, otherwise your modifications can be destroyed during your next update!

File	Description
<u>/etc/profile.local</u>	Use this file if you extend <u>/etc/profile</u>
<u>/etc/profile.d/</u>	Contains system-wide configuration files for specific programs
<u>~/.profile</u>	Insert user specific configuration for login shells here

TABLE 1.2: BASH CONFIGURATION FILES FOR NON-LOGIN SHELLS

<u>/etc/bash.bashrc</u>	Do not modify this file, otherwise your modifications can be destroyed during your next update!
<u>/etc/bash.bashrc.local</u>	Use this file to insert your system-wide modifications for Bash only
<u>~/.bashrc</u>	Insert user specific configuration here

Additionally, Bash uses some more files:

TABLE 1.3: SPECIAL FILES FOR BASH

File	Description
<u>~/.bash_history</u>	Contains a list of all commands you have been typing
<u>~/.bash_logout</u>	Executed when logging out

1.1.2 The Directory Structure

The following table provides a short overview of the most important higher-level directories that you find on a Linux system. Find more detailed information about the directories and important subdirectories in the following list.

TABLE 1.4: OVERVIEW OF A STANDARD DIRECTORY TREE

Directory	Contents
<u>/</u>	Root directory—the starting point of the directory tree.
<u>/bin</u>	Essential binary files, such as commands that are needed by both the system administrator and normal users. Usually also contains the shells, such as Bash.
<u>/boot</u>	Static files of the boot loader.
<u>/dev</u>	Files needed to access host-specific devices.
<u>/etc</u>	Host-specific system configuration files.
<u>/home</u>	Holds the home directories of all users who have accounts on the system. However, <u>root</u> 's home directory is not located in <u>/home</u> but in <u>/root</u> .
<u>/lib</u>	Essential shared libraries and kernel modules.
<u>/media</u>	Mount points for removable media.
<u>/mnt</u>	Mount point for temporarily mounting a file system.
<u>/opt</u>	Add-on application software packages.
<u>/root</u>	Home directory for the superuser <u>root</u> .
<u>/sbin</u>	Essential system binaries.
<u>/srv</u>	Data for services provided by the system.
<u>/tmp</u>	Temporary files.
<u>/usr</u>	Secondary hierarchy with read-only data.
<u>/var</u>	Variable data such as log files.
<u>/windows</u>	Only available if you have both Microsoft Windows* and Linux installed on your system. Contains the Windows data.

The following list provides more detailed information and gives some examples of which files and subdirectories can be found in the directories:

/bin

Contains the basic shell commands that may be used both by root and by other users. These commands include ls, mkdir, cp, mv, rm and rmdir. /bin also contains Bash, the default shell in SUSE Linux Enterprise Server.

/boot

Contains data required for booting, such as the boot loader, the kernel, and other data that is used before the kernel begins executing user-mode programs.

/dev

Holds device files that represent hardware components.

/etc

Contains local configuration files that control the operation of programs like the X Window System. The /etc/init.d subdirectory contains LSB init scripts that can be executed during the boot process.

/home/username

Holds the private data of every user who has an account on the system. The files located here can only be modified by their owner or by the system administrator. By default, your e-mail directory and personal desktop configuration are located here in the form of hidden files and directories, such as .gconf/ and .config.



Note: Home Directory in a Network Environment

If you are working in a network environment, your home directory may be mapped to a directory in the file system other than /home.

/lib

Contains the essential shared libraries needed to boot the system and to run the commands in the root file system. The Windows equivalent for shared libraries are DLL files.

/media

Contains mount points for removable media, such as CD-ROMs, flash disks, and digital cameras (if they use USB). /media generally holds any type of drive except the hard disk of your system. When your removable medium has been inserted or connected to the system and has been mounted, you can access it from here.

/mnt

This directory provides a mount point for a temporarily mounted file system. root may mount file systems here.

/opt

Reserved for the installation of third-party software. Optional software and larger add-on program packages can be found here.

/root

Home directory for the root user. The personal data of root is located here.

/run

A tmpfs directory used by systemd and various components. /var/run is a symbolic link to /run.

/sbin

As the s indicates, this directory holds utilities for the superuser. /sbin contains the binaries essential for booting, restoring and recovering the system in addition to the binaries in /bin.

/srv

Holds data for services provided by the system, such as FTP and HTTP.

/tmp

This directory is used by programs that require temporary storage of files.



Important: Cleaning up /tmp at Boot Time

Data stored in /tmp are not guaranteed to survive a system reboot. It depends, for example, on settings in /etc/sysconfig/cron.

/usr

/usr has nothing to do with users, but is the acronym for Unix system resources. The data in /usr is static, read-only data that can be shared among various hosts compliant with the Filesystem Hierarchy Standard (FHS). This directory contains all application programs including the graphical desktops such as GNOME and establishes a secondary hierarchy in the file system. /usr holds several subdirectories, such as /usr/bin, /usr/sbin, /usr/local, and /usr/share/doc.

/usr/bin

Contains generally accessible programs.

/usr/sbin

Contains programs reserved for the system administrator, such as repair functions.

/usr/local

In this directory the system administrator can install local, distribution-independent extensions.

/usr/share/doc

Holds various documentation files and the release notes for your system. In the manual subdirectory find an online version of this manual. If more than one language is installed, this directory may contain versions of the manuals for different languages.

Under packages find the documentation included in the software packages installed on your system. For every package, a subdirectory /usr/share/doc/packages/package-name is created that often holds README files for the package and sometimes examples, configuration files or additional scripts.

If HOWTOs are installed on your system /usr/share/doc also holds the howto subdirectory in which to find additional documentation on many tasks related to the setup and operation of Linux software.

/var

Whereas /usr holds static, read-only data, /var is for data which is written during system operation and thus is variable data, such as log files or spooling data. For an overview of the most important log files you can find under /var/log/, refer to *Table 38.1, "Log Files"*.

1.2 Writing Shell Scripts

Shell scripts are a convenient way of doing all sorts of tasks: collecting data, searching for a word or phrase in a text and many other useful things. The following example shows a small shell script that prints a text:

EXAMPLE 1.1: A SHELL SCRIPT PRINTING A TEXT

```
#!/bin/sh ❶  
# Output the following line: ❷  
echo "Hello World" ❸
```

- ❶ The first line begins with the *Shebang* characters (`#!`) which is an indicator that this file is a script. The script is executed with the specified interpreter after the Shebang, in this case `/bin/sh`.
- ❷ The second line is a comment beginning with the hash sign. It is recommended to comment difficult lines to remember what they do.
- ❸ The third line uses the built-in command `echo` to print the corresponding text.

Before you can run this script you need some prerequisites:

1. Every script should contain a Shebang line (this is already the case with our example above.) If a script does not have this line, you need to call the interpreter manually.
2. You can save the script wherever you want. However, it is a good idea to save it in a directory where the shell can find it. The search path in a shell is determined by the environment variable `PATH`. Usually a normal user does not have write access to `/usr/bin`. Therefore it is recommended to save your scripts in the users' directory `~/bin/`. The above example gets the name `hello.sh`.
3. The script needs executable permissions. Set the permissions with the following command:

```
chmod +x ~/bin/hello.sh
```

If you have fulfilled all of the above prerequisites, you can execute the script in the following ways:

1. **As Absolute Path.** The script can be executed with an absolute path. In our case, it is `~/bin/hello.sh`.
2. **Everywhere.** If the `PATH` environment variable contains the directory where the script is located, you can execute the script with `hello.sh`.

1.3 Redirecting Command Events

Each command can use three channels, either for input or output:

- **Standard Output.** This is the default output channel. Whenever a command prints something, it uses the standard output channel.
- **Standard Input.** If a command needs input from users or other commands, it uses this channel.
- **Standard Error.** Commands use this channel for error reporting.

To redirect these channels, there are the following possibilities:

Command > File

Saves the output of the command into a file, an existing file will be deleted. For example, the **ls** command writes its output into the file listing.txt:

```
ls > listing.txt
```

Command >> File

Appends the output of the command to a file. For example, the **ls** command appends its output to the file listing.txt:

```
ls >> listing.txt
```

Command < File

Reads the file as input for the given command. For example, the **read** command reads in the content of the file into the variable:

```
read a < foo
```

Command1 | Command2

Redirects the output of the left command as input for the right command. For example, the **cat** command outputs the content of the /proc/cpuinfo file. This output is used by **grep** to filter only those lines which contain cpu:

```
cat /proc/cpuinfo | grep cpu
```

Every channel has a *file descriptor*: 0 (zero) for standard input, 1 for standard output and 2 for standard error. It is allowed to insert this file descriptor before a `<` or `>` character. For example, the following line searches for a file starting with `foo`, but suppresses its errors by redirecting it to `/dev/null`:

```
find / -name "foo*" 2>/dev/null
```

1.4 Using Aliases

An alias is a shortcut definition of one or more commands. The syntax for an alias is:

```
alias NAME=DEFINITION
```

For example, the following line defines an alias `lt` which outputs a long listing (option `-l`), sorts it by modification time (`-t`) and prints it in reverse order while sorting (`-r`):

```
alias lt='ls -ltr'
```

To view all alias definitions, use `alias`. Remove your alias with `unalias` and the corresponding alias name.

1.5 Using Variables in Bash

A shell variable can be global or local. Global variables, or environment variables, can be accessed in all shells. In contrast, local variables are visible in the current shell only.

To view all environment variables, use the `printenv` command. If you need to know the value of a variable, insert the name of your variable as an argument:

```
printenv PATH
```

A variable, be it global or local, can also be viewed with `echo`:

```
echo $PATH
```

To set a local variable, use a variable name followed by the equal sign, followed by the value:

```
PROJECT="SLED"
```

Do not insert spaces around the equal sign, otherwise you get an error. To set an environment variable, use **export**:

```
export NAME="tux"
```

To remove a variable, use **unset**:

```
unset NAME
```

The following table contains some common environment variables which can be used in you shell scripts:

TABLE 1.5: USEFUL ENVIRONMENT VARIABLES

<u>HOME</u>	the home directory of the current user
<u>HOST</u>	the current host name
<u>LANG</u>	when a tool is localized, it uses the language from this environment variable. English can also be set to <u>C</u>
<u>PATH</u>	the search path of the shell, a list of directories separated by colon
<u>PS1</u>	specifies the normal prompt printed before each command
<u>PS2</u>	specifies the secondary prompt printed when you execute a multi-line command
<u>PWD</u>	current working directory
<u>USER</u>	the current user

1.5.1 Using Argument Variables

For example, if you have the script **foo.sh** you can execute it like this:

```
foo.sh "Tux Penguin" 2000
```

To access all the arguments which are passed to your script, you need positional parameters. These are \$1 for the first argument, \$2 for the second, and so on. You can have up to nine parameters. To get the script name, use \$0.

The following script **foo.sh** prints all arguments from 1 to 4:

```
#!/bin/sh
echo \"$1\" \"$2\" \"$3\" \"$4\"
```

If you execute this script with the above arguments, you get:

```
"Tux Penguin" "2000" "" ""
```

1.5.2 Using Variable Substitution

Variable substitutions apply a pattern to the content of a variable either from the left or right side. The following list contains the possible syntax forms:

\${VAR#pattern}

removes the shortest possible match from the left:

```
file=/home/tux/book/book.tar.bz2
echo ${file#*/}
home/tux/book/book.tar.bz2
```

\${VAR##pattern}

removes the longest possible match from the left:

```
file=/home/tux/book/book.tar.bz2
echo ${file##*/}
book.tar.bz2
```

\${VAR%pattern}

removes the shortest possible match from the right:

```
file=/home/tux/book/book.tar.bz2
echo ${file%.*}
/home/tux/book/book.tar
```

`${VAR%%pattern}`

removes the longest possible match from the right:

```
file=/home/tux/book/book.tar.bz2
echo ${file%%.*}
/home/tux/book/book
```

`${VAR/pattern_1/pattern_2}`

substitutes the content of VAR from the pattern_1 with pattern_2:

```
file=/home/tux/book/book.tar.bz2
echo ${file/tux/wilber}
/home/wilber/book/book.tar.bz2
```

1.6 Grouping and Combining Commands

Shells allow you to concatenate and group commands for conditional execution. Each command returns an exit code which determines the success or failure of its operation. If it is 0 (zero) the command was successful, everything else marks an error which is specific to the command.

The following list shows, how commands can be grouped:

Command1 ; Command2

executes the commands in sequential order. The exit code is not checked. The following line displays the content of the file with cat and then prints its file properties with ls regardless of their exit codes:

```
cat filelist.txt ; ls -l filelist.txt
```

Command1 && Command2

runs the right command, if the left command was successful (logical AND). The following line displays the content of the file and prints its file properties only, when the previous command was successful (compare it with the previous entry in this list):

```
cat filelist.txt && ls -l filelist.txt
```

Command1 || Command2

runs the right command, when the left command has failed (logical OR). The following line creates only a directory in /home/wilber/bar when the creation of the directory in /home/tux/foo has failed:

```
mkdir /home/tux/foo || mkdir /home/wilber/bar
```

funcname(){ ... }

creates a shell function. You can use the positional parameters to access its arguments. The following line defines the function hello to print a short message:

```
hello() { echo "Hello $1"; }
```

You can call this function like this:

```
hello Tux
```

which prints:

```
Hello Tux
```

1.7 Working with Common Flow Constructs

To control the flow of your script, a shell has while, if, for and case constructs.

1.7.1 The if Control Command

The if command is used to check expressions. For example, the following code tests whether the current user is Tux:

```
if test $USER = "tux"; then
    echo "Hello Tux."
else
    echo "You are not Tux."
fi
```

The test expression can be as complex or simple as possible. The following expression checks if the file `foo.txt` exists:

```
if test -e /tmp/foo.txt ; then
    echo "Found foo.txt"
fi
```

The test expression can also be abbreviated in angled brackets:

```
if [ -e /tmp/foo.txt ] ; then
    echo "Found foo.txt"
fi
```

Find more useful expressions at <http://www.cyberciti.biz/nixcraft/linux/docs/uniqlinuxfeatures/lsst/ch03sec02.html>.

1.7.2 Creating Loops With the **for** Command

The **for** loop allows you to execute commands to a list of entries. For example, the following code prints some information about PNG files in the current directory:

```
for i in *.png; do
    ls -l $i
done
```

1.8 For More Information

Important information about Bash is provided in the man pages `man bash`. More about this topic can be found in the following list:

- <http://tldp.org/LDP/Bash-Beginners-Guide/html/index.html>—Bash Guide for Beginners
- <http://tldp.org/HOWTO/Bash-Prog-Intro-HOWTO.html>—BASH Programming - Introduction HOW-TO
- <http://tldp.org/LDP/abs/html/index.html>—Advanced Bash-Scripting Guide
- <http://www.grymoire.com/Unix/Sh.html>—Sh - the Bourne Shell

2 sudo

Many commands and system utilities need to be run as root to modify files and/or perform tasks that only the super user is allowed to. For security reasons and to avoid accidentally running dangerous commands, it is generally advisable not to log in directly as root. Instead, it is recommended to work as a normal, unprivileged user and use the sudo command to run commands with elevated privileges.

On SUSE Linux Enterprise Server, sudo is configured by default to work similarly to su. However, sudo offers the possibility to allow users to run commands with privileges of any other user in a highly configurable manner. This can be used to assign roles with specific privileges to certain users and groups. It is for example possible to allow members of the group users to run a command with the privileges of wilber. Access to the command can be further restricted by, for example, forbidding to specify any command options. While su always requires the root password for authentication with PAM, sudo can be configured to authenticate with your own credentials. This increases security by not having to share the root password. For instance, you can allow members of the group users to run a command **frobnicate** as wilber, with the restriction that no arguments are specified. This can be used to assign roles with specific abilities to certain users and groups.

2.1 Basic **sudo** Usage

sudo is simple to use, yet very powerful.

2.1.1 Running a Single Command

Logged in as normal user, you can run any command as root by adding sudo before it. It will prompt for the root password and, if authenticated successfully, run the command as root:

```
tux > id -un ❶
tux
tux > sudo id -un
root's password: ❷
root
tux > id -un
tux ❸
tux > sudo id -un
```

4

root

- ❶ The `id -un` command prints the login name of the current user.
- ❷ The password is not shown during input, neither as clear text nor as bullets.
- ❸ Only commands started with `sudo` are run with elevated privileges. If you run the same command without the `sudo` prefix, it is run with the privileges of the current user again.
- ❹ For a limited amount of time, you do not need to enter the `root` password again.



Tip: I/O Redirection

I/O redirection does not work as you would probably expect:

```
tux > sudo echo s > /proc/sysrq-trigger
bash: /proc/sysrq-trigger: Permission denied
tux > sudo cat < /proc/l/maps
bash: /proc/l/maps: Permission denied
```

Only the `echo` / `cat` binary is run with elevated privileges, while the redirection is performed by the user's shell with user privileges. You can either start a shell like in [Section 2.1.2, "Starting a Shell"](#) or use the `dd` utility instead:

```
echo s | sudo dd of=/proc/sysrq-trigger
sudo dd if=/proc/l/maps | cat
```

2.1.2 Starting a Shell

Having to add `sudo` before every command can be cumbersome. While you could specify a shell as a command `sudo bash`, it is recommended to rather use one of the built-in mechanisms to start a shell:

`sudo -s (<command>)`

Starts a shell specified by the `SHELL` environment variable or the target user's default shell. If a command is given, it is passed to the shell (with the `-c` option), else the shell is run in interactive mode.

```
tux:~ > sudo -i
```

```
root's password:
root:/home/tux # exit
tux:~ >
```

`sudo -i (<command>)`

Like `-s`, but starts the shell as login shell. This means that the shell's start-up files (`.profile` etc.) are processed and the current working directory is set to the target user's home directory.

```
tux:~ > sudo -i
root's password:
root:~ # exit
tux:~ >
```

2.1.3 Environment Variables

By default, `sudo` does not propagate environment variables:

```
tux > ENVVAR=test env | grep ENVVAR
ENVVAR=test
tux > ENVVAR=test sudo env | grep ENVVAR
root's password:
❶
tux >
```

- ❶ The empty output shows that the environment variable `ENVVAR` did not exist in the context of the command run with `sudo`.

This behavior can be changed by the `env_reset` option, see [Table 2.1, "Useful Flags and Options"](#).

2.2 Configuring `sudo`

`sudo` is a very flexible tool with extensive configuration.



Note: Locked yourself out of `sudo`

If you locked yourself out of `sudo` by mistake, you can use `su -` and the `root` password to get a root shell and run `visudo` to fix the error.

2.2.1 Editing the Configuration Files

The main policy configuration file for **sudo** is `/etc/sudoers`. As it is possible to lock yourself out of the system if there are any errors in that file, it is strongly recommended to use **visudo** for editing. It will prevent simultaneous changes to the opened file and check for syntax errors before saving.

Despite its name, you can also use editors other than `vi` by setting the `EDITOR` environment variable, for example:

```
sudo EDITOR=/usr/bin/nano visudo
```

However, the `/etc/sudoers` file itself is supplied by the system packages and modifications may break on updates. Therefore, it is recommended to put custom configuration into files in the `/etc/sudoers.d/` directory. Any file in there is automatically included. To create or edit a file in that subdirectory, run:

```
sudo visudo -f /etc/sudoers.d/NAME
```

Alternatively with a different editor (for example **nano**):

```
sudo EDITOR=/usr/bin/nano visudo -f /etc/sudoers.d/NAME
```



Note: Ignored Files in `/etc/sudoers.d`

The `#includedir` command in `/etc/sudoers`, used for `/etc/sudoers.d`, ignores files that end in `~` (tilde) or contain a `.` (dot).

For more information on the **visudo** command, run `man 8 visudo`.

2.2.2 Basic sudoers Configuration Syntax

In the sudoers configuration files, there are two types of options: strings and flags. While strings can contain any value, flags can be turned either ON or OFF. The most important syntax constructs for sudoers configuration files are:

```
# Everything on a line after a # gets ignored ❶  
Defaults !insults # Disable the insults flag ❷
```

```
Defaults env_keep += "DISPLAY HOME" # Add DISPLAY and HOME to env_keep
tux ALL = NOPASSWD: /usr/bin/frobnicate, PASSWD: /usr/bin/journalctl ❸
```

- ❶ There are two exceptions: `#include` and `#includedir` are normal commands. Followed by digits, it specifies a UID.
- ❷ Remove the `!` to set the specified flag to ON.
- ❸ See [Section 2.2.3, “Rules in sudoers”](#).

TABLE 2.1: USEFUL FLAGS AND OPTIONS

Option name	Description	Example
<u>targetpw</u>	This flag controls whether the invoking user is required to enter the password of the target user (ON) (for example <u>root</u>) or the invoking user (OFF).	Defaults targetpw # Turn targetpw flag ON
<u>rootpw</u>	If set, sudo will prompt for the <u>root</u> password instead of the target user's or the invoker's. The default is OFF.	Defaults !rootpw # Turn rootpw flag OFF
<u>env_reset</u>	If set, sudo constructs a minimal environment with only <u>TERM</u> , <u>PATH</u> , <u>HOME</u> , <u>MAIL</u> , <u>SHELL</u> , <u>LOGNAME</u> , <u>USER</u> , <u>USERNAME</u> , and <u>SUDO_*</u> set. Additionally, variables listed in <u>env_keep</u> get imported from the calling environment. The default is ON.	Defaults env_reset # Turn env_reset flag ON
<u>env_keep</u>	List of environment variables to keep when the <u>env_reset</u> flag is ON.	# Set env_keep to contain EDITOR and PROMPT

Option name	Description	Example
		<pre>Defaults env_keep = "EDITOR PROMPT" Defaults env_keep += "JRE_HOME" # Add JRE_HOME Defaults env_keep -= "JRE_HOME" # Remove JRE_HOME</pre>
<u>env_delete</u>	List of environment variables to remove when the <u>env_reset</u> flag is OFF.	<pre># Set env_delete to contain EDITOR and PROMPT Defaults env_delete = "EDITOR PROMPT" Defaults env_delete += "JRE_HOME" # Add JRE_HOME Defaults env_delete - = "JRE_HOME" # Remove JRE_HOME</pre>

The Defaults token can also be used to create aliases for a collection of users, hosts, and commands. Furthermore, it is possible to apply an option only to a specific set of users.

For detailed information about the /etc/sudoers configuration file, consult man 5 sudoers.

2.2.3 Rules in sudoers

Rules in the sudoers configuration can be very complex, so this section will only cover the basics. Each rule follows the basic scheme ([] marks optional parts):

#Who	Where	As whom	Tag	What
User_List	Host_List	= [(User_List)]	[NOPASSWD: PASSWD:]	Cmnd_List

SYNTAX FOR SUDOERS RULES

User_List

One or more (separated by ,) identifiers: Either a user name, a group in the format %GROUPNAME or a user ID in the format #UID. Negation can be performed with a ! prefix.

Host_List

One or more (separated by ,) identifiers: Either a (fully qualified) host name or an IP address. Negation can be performed with a ! prefix. ALL is the usual choice for Host_List.

NOPASSWD: | PASSWD:

The user will not be prompted for a password when running commands matching CMDSPEC after NOPASSWD:.

PASSWD is the default, it only needs to be specified when both are on the same line:

```
tux ALL = PASSWD: /usr/bin/foo, NOPASSWD: /usr/bin/bar
```

Cmnd_List

One or more (separated by ,) specifiers: A path to an executable, followed by allowed arguments or nothing.

```
/usr/bin/foo      # Anything allowed
/usr/bin/foo bar  # Only "/usr/bin/foo bar" allowed
/usr/bin/foo ""   # No arguments allowed
```

ALL can be used as User_List, Host_List, and Cmnd_List.

A rule that allows tux to run all commands as root without entering a password:

```
tux ALL = NOPASSWD: ALL
```

A rule that allows tux to run systemctl restart apache2:

```
tux ALL = /usr/bin/systemctl restart apache2
```

A rule that allows tux to run wall as admin with no arguments:

```
tux ALL = (admin) /usr/bin/wall ""
```



Warning: Dangerous constructs

Constructs of the kind

```
ALL ALL = ALL
```

must not be used without Defaults targetpw, otherwise anyone can run commands as root.

2.3 Common Use Cases

Although the default configuration is often sufficient for simple setups and desktop environments, custom configurations can be very useful.

2.3.1 Using **sudo** without root Password

In cases with special restrictions (“user X can only run command Y as root”) it is not possible. In other cases, it is still favorable to have some kind of separation. By convention, members of the group wheel can run all commands with sudo as root.

1. Add yourself to the wheel group

If your user account is not already member of the wheel group, add it by running **sudo usermod -a -G wheel USERNAME** and logging out and in again. Verify that the change was successful by running **groups USERNAME**.

2. Make authentication with the invoking user's password the default.

Create the file /etc/sudoers.d/userpw with **visudo** (see [Section 2.2.1, “Editing the Configuration Files”](#)) and add:

```
Defaults !targetpw
```

3. Select a new default rule.

Depending on whether you want users to re-enter their passwords, uncomment the specific line in /etc/sudoers and comment out the default rule.

```
## Uncomment to allow members of group wheel to execute any command
# %wheel ALL=(ALL) ALL

## Same thing without a password
# %wheel ALL=(ALL) NOPASSWD: ALL
```

4. Make the default rule more restrictive

Comment out or remove the allow-everything rule in /etc/sudoers:

```
ALL    ALL=(ALL) ALL    # WARNING! Only use this together with 'Defaults
targetpw'!
```




Warning: Dangerous rule in sudoers

Do not forget this step, otherwise *any* user can execute *any* command as root !

5. Test the configuration

Try to run sudo as member and non-member of wheel.

```
tux:~ > groups
users wheel
tux:~ > sudo id -un
tux's password:
root
wilber:~ > groups
users
wilber:~ > sudo id -un
wilber is not in the sudoers file. This incident will be reported.
```

2.3.2 Using **sudo** with X.Org Applications

When starting graphical applications with sudo, you will encounter the following error:

```
tux > sudo xterm
xterm: Xt error: Can't open display: %s
xterm: DISPLAY is not set
```

YaST will pick the ncurses interface instead of the graphical one.

To use X.Org in applications started with sudo, the environment variables DISPLAY and XAUTHORITY need to be propagated. To configure this, create the file /etc/sudoers.d/xorg, (see [Section 2.2.1, "Editing the Configuration Files"](#)) and add the following line:

```
Defaults env_keep += "DISPLAY XAUTHORITY"
```

If not set already, set the XAUTHORITY variable as follows:

```
export XAUTHORITY=~/.Xauthority
```

Now X.Org applications can be run as usual:

```
sudo yast2
```

2.4 More Information

A quick overview about the available command line switches can be retrieved by `sudo --help`. An explanation and other important information can be found in the man page: `man 8 sudo`, while the configuration is documented in `man 5 sudoers`.

3 YaST Online Update

SUSE offers a continuous stream of software security updates for your product. By default, the update applet is used to keep your system up-to-date. Refer to *Book “Deployment Guide”, Chapter 12 “Installing or Removing Software”, Section 12.4 “Keeping the System Up-to-date”* for further information on the update applet. This chapter covers the alternative tool for updating software packages: YaST Online Update.

The current patches for SUSE® Linux Enterprise Server are available from an update software repository. If you have registered your product during the installation, an update repository is already configured. If you have not registered SUSE Linux Enterprise Server, you can do so by starting the *Product Registration* in YaST. Alternatively, you can manually add an update repository from a source you trust. To add or remove repositories, start the Repository Manager with *Software > Software Repositories* in YaST. Learn more about the Repository Manager in *Book “Deployment Guide”, Chapter 12 “Installing or Removing Software”, Section 12.3 “Managing Software Repositories and Services”*.



Note: Error on Accessing the Update Catalog

If you are not able to access the update catalog, this might be because of an expired subscription. Normally, SUSE Linux Enterprise Server comes with a one-year or three-year subscription, during which you have access to the update catalog. This access will be denied after the subscription ends.

In case of an access denial to the update catalog you will see a warning message with a recommendation to visit the SUSE Customer Center and check your subscription. The SUSE Customer Center is available at <https://scc.suse.com/> .

SUSE provides updates with different relevance levels:

Security Updates

Fix severe security hazards and should always be installed.

Recommended Updates

Fix issues that could compromise your computer.

Optional Updates

Fix non-security relevant issues or provide enhancements.

3.1 The Online Update Dialog

To open the YaST *Online Update* dialog, start YaST and select *Software > Online Update*. Alternatively, start it from the command line with **yast2 online_update**.

The *Online Update* window consists of four sections.

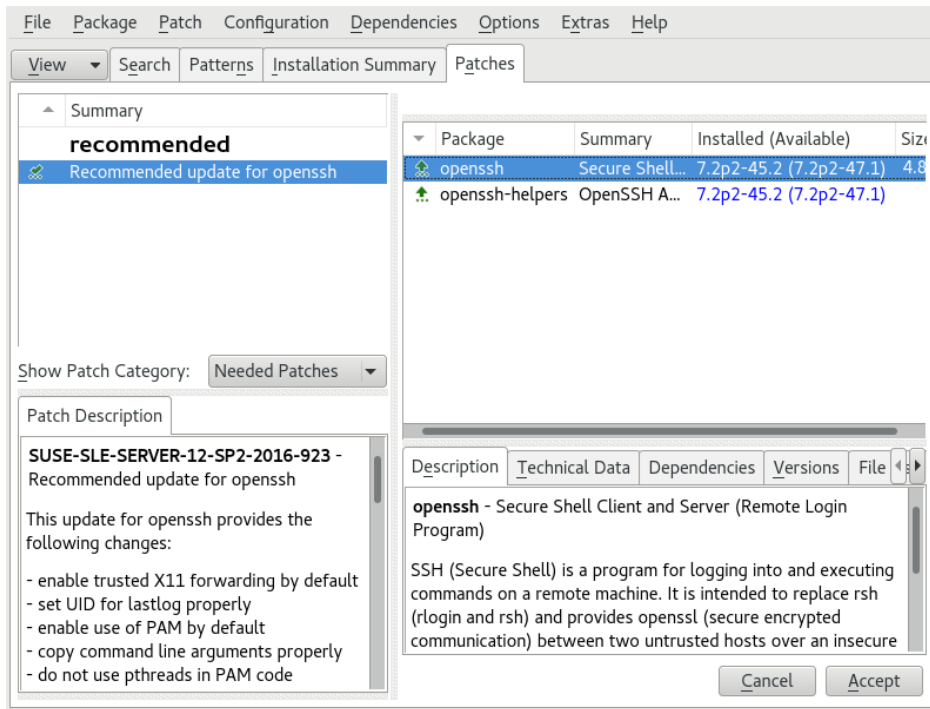


FIGURE 3.1: YAST ONLINE UPDATE

The *Summary* section on the left lists the available patches for SUSE Linux Enterprise Server. The patches are sorted by security relevance: security, recommended, and optional. You can change the view of the *Summary* section by selecting one of the following options from *Show Patch Category*:

Needed Patches (default view)

Non-installed patches that apply to packages installed on your system.

Unneeded Patches

Patches that either apply to packages not installed on your system, or patches that have requirements which have already been fulfilled (because the relevant packages have already been updated from another source).

All Patches

All patches available for SUSE Linux Enterprise Server.

Each list entry in the *Summary* section consists of a symbol and the patch name. For an overview of the possible symbols and their meaning, press **Shift-F1**. Actions required by Security and Recommended patches are automatically preset. These actions are *Autoinstall*, *Autoupdate* and *Autodelete*.

If you install an up-to-date package from a repository other than the update repository, the requirements of a patch for this package may be fulfilled with this installation. In this case a check mark is displayed in front of the patch summary. The patch will be visible in the list until you mark it for installation. This will in fact not install the patch (because the package already is up-to-date), but mark the patch as having been installed.

Select an entry in the *Summary* section to view a short *Patch Description* at the bottom left corner of the dialog. The upper right section lists the packages included in the selected patch (a patch can consist of several packages). Click an entry in the upper right section to view details about the respective package that is included in the patch.

3.2 Installing Patches

The YaST Online Update dialog allows you to either install all available patches at once or to manually select the patches that you want to apply to your system. You may also revert patches that have been applied to the system.

By default, all new patches (except optional ones) that are currently available for your system are already marked for installation. They will be applied automatically once you click *Accept* or *Apply*. If one or multiple patches require a system reboot, you will be notified about this before the patch installation starts. You can then either decide to continue with the installation of the selected patches, skip the installation of all patches that need rebooting and install the rest, or go back to the manual patch selection.

PROCEDURE 3.1: APPLYING PATCHES WITH YAST ONLINE UPDATE

1. Start YaST and select *Software > Online Update*.
2. To automatically apply all new patches (except optional ones) that are currently available for your system, proceed with *Apply* or *Accept* to start the installation of the preselected patches.

3. First modify the selection of patches that you want to apply:

- a. Use the respective filters and views that the interface provides. For details, refer to *Section 3.1, “The Online Update Dialog”*.
- b. Select or deselect patches according to your needs and wishes by right-clicking the patch and choosing the respective action from the context menu.



Important: Always Apply Security Updates

Do not deselect any security-related patches without a very good reason. These patches fix severe security hazards and prevent your system from being exploited.

- c. Most patches include updates for several packages. If you want to change actions for single packages, right-click a package in the package view and choose an action.
 - d. To confirm your selection and apply the selected patches, proceed with *Apply* or *Accept*.
4. After the installation is complete, click *Finish* to leave the YaST *Online Update*. Your system is now up-to-date.

3.3 Automatic Online Update

YaST also offers the possibility to set up an automatic update with daily, weekly or monthly schedule. To use the respective module, you need to install the yast2-online-update-configuration package first.

By default, updates are downloaded as delta RPMs. Since rebuilding RPM packages from delta RPMs is a memory- and processor-consuming task, certain setups or hardware configurations might require you to disable the use of delta RPMs for the sake of performance.

Some patches, such as kernel updates or packages requiring license agreements, require user interaction, which would cause the automatic update procedure to stop. You can configure to skip patches that require user interaction.

PROCEDURE 3.2: CONFIGURING THE AUTOMATIC ONLINE UPDATE

1. After installation, start YaST and select *Software > Online Update Configuration*.

Alternatively, start the module with yast2 online_update_configuration from the command line.

2. Activate *Automatic Online Update*.
3. Choose the update interval: *Daily*, *Weekly*, or *Monthly*.
4. To automatically accept any license agreements, activate *Agree with Licenses*.
5. Select if you want to *Skip Interactive Patches* in case you want the update procedure to proceed fully automatically.

Important: Skipping Patches

If you select to skip any packages that require interaction, run a manual *Online Update* occasionally to install those patches, too. Otherwise you might miss important patches.

6. To automatically install all packages recommended by updated packages, activate *Include Recommended Packages*.
7. To disable the use of delta RPMs (for performance reasons), deactivate *Use Delta RPMs*.
8. To filter the patches by category (such as security or recommended), activate *Filter by Category* and add the appropriate patch categories from the list. Only patches of the selected categories will be installed. Others will be skipped.
9. Confirm your configuration with *OK*.

The automatic online update does not automatically restart the system afterward. If there are package updates that require a system reboot, you need to do this manually.

4 YaST in Text Mode

This section is intended for system administrators and experts who do not run an X server on their systems and depend on the text-based installation tool. It provides basic information about starting and operating YaST in text mode.

YaST in text mode uses the ncurses library to provide an easy pseudo-graphical user interface. The ncurses library is installed by default. The minimum supported size of the terminal emulator in which to run YaST is 80x25 characters.

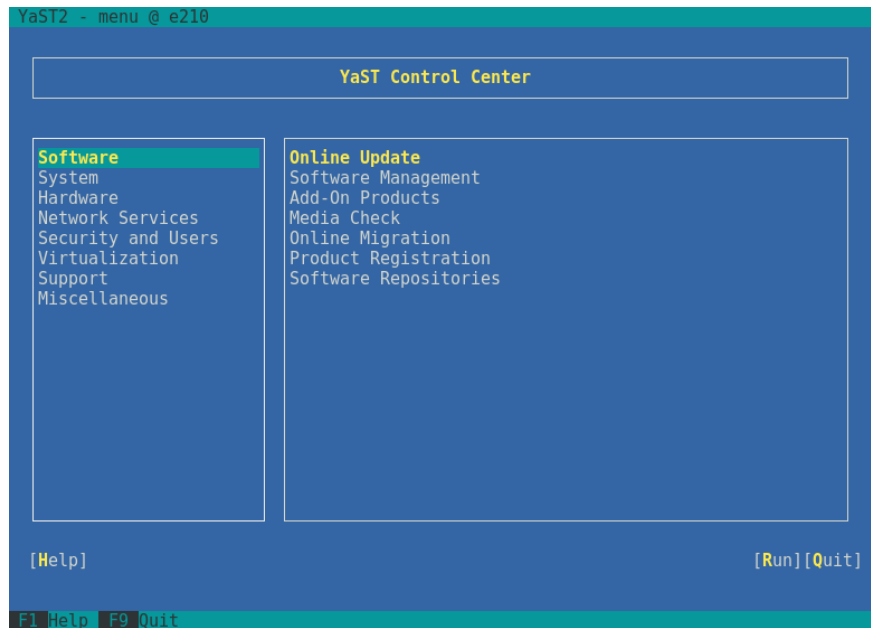


FIGURE 4.1: MAIN WINDOW OF YAST IN TEXT MODE

When you start YaST in text mode, the YaST control center appears (see [Figure 4.1](#)). The main window consists of three areas. The left frame features the categories to which the various modules belong. This frame is active when YaST is started and therefore it is marked by a bold white border. The active category is selected. The right frame provides an overview of the modules available in the active category. The bottom frame contains the buttons for *Help* and *Quit*.

When you start the YaST control center, the category *Software* is selected automatically. Use and to change the category. To select a module from the category, activate the right frame with and then use and to select the module. Keep the arrow keys pressed to scroll through the list of available modules. The selected module is highlighted. Press to start the active module.

Various buttons or selection fields in the module contain a highlighted letter (yellow by default). Use `Alt`-`highlighted_letter` to select a button directly instead of navigating there with `→|`. Exit the YaST control center by pressing `Alt`-`Q` or by selecting *Quit* and pressing `Enter`.



Tip: Refreshing YaST Dialogs

If a YaST dialog gets corrupted or distorted (for example, while resizing the window), press `Ctrl`-`L` to refresh and restore its contents.

4.1 Navigation in Modules

The following description of the control elements in the YaST modules assumes that all function keys and `Alt` key combinations work and are not assigned to different global functions. Read [Section 4.2, “Restriction of Key Combinations”](#) for information about possible exceptions.

Navigation among Buttons and Selection Lists

Use `→|` to navigate among the buttons and frames containing selection lists. To navigate in reverse order, use `Alt`-`→|` or `Shift`-`→|` combinations.

Navigation in Selection Lists

Use the arrow keys (`↑` and `↓`) to navigate among the individual elements in an active frame containing a selection list. If individual entries within a frame exceed its width, use `Shift`-`→` or `Shift`-`←` to scroll horizontally to the right and left. Alternatively, use `Ctrl`-`E` or `Ctrl`-`A`. This combination can also be used if using `→` or `←` results in changing the active frame or the current selection list, as in the control center.

Buttons, Radio Buttons, and Check Boxes

To select buttons with empty square brackets (check boxes) or empty parentheses (radio buttons), press `Space` or `Enter`. Alternatively, radio buttons and check boxes can be selected directly with `Alt`-`highlighted_letter`. In this case, you do not need to confirm with `Enter`. If you navigate to an item with `→|`, press `Enter` to execute the selected action or activate the respective menu item.

Function Keys

The F keys (**F1** through **F12**) enable quick access to the various buttons. Available function key combinations (**Fx**) are shown in the bottom line of the YaST screen. Which function keys are actually mapped to which buttons depend on the active YaST module, because the different modules offer different buttons (*Details*, *Info*, *Add*, *Delete*, etc.). Use **F10** for *Accept*, *OK*, *Next*, and *Finish*. Press **F1** to access the YaST help.

Using Navigation Tree in ncurses Mode

Some YaST modules use a navigation tree in the left part of the window to select configuration dialogs. Use the arrow keys (**↑** and **↓**) to navigate in the tree. Use **Space** to open or close tree items. In ncurses mode, **Enter** must be pressed after a selection in the navigation tree to show the selected dialog. This is an intentional behavior to save time consuming redraws when browsing through the navigation tree.

Selecting Software in the Software Installation Module

Use the filters on the left side to limit the amount of displayed packages. Installed packages are marked with the letter **i**. To change the status of a package, press **Space** or **Enter**. Alternatively, use the *Actions* menu to select the needed status change (install, delete, update, taboo or lock).

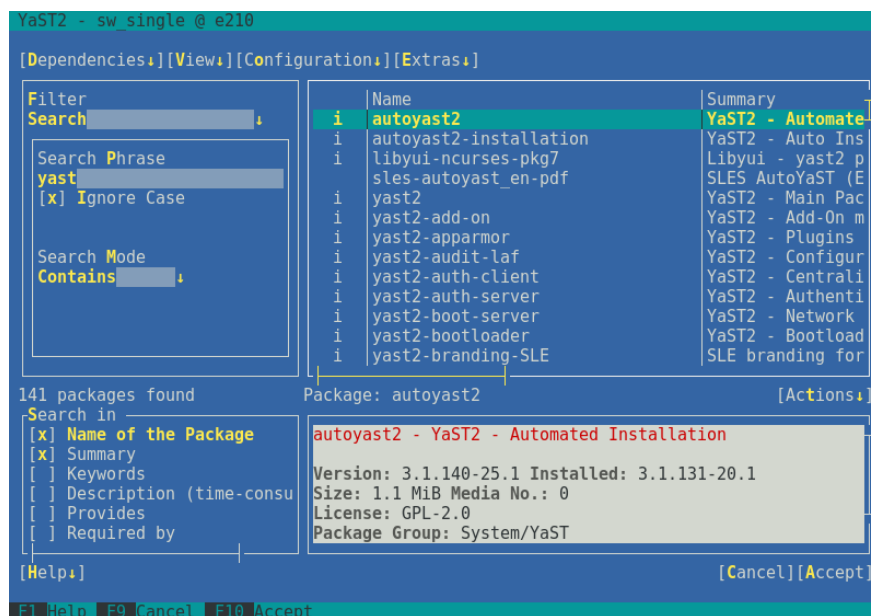


FIGURE 4.2: THE SOFTWARE INSTALLATION MODULE

4.2 Restriction of Key Combinations

If your window manager uses global **Alt** combinations, the **Alt** combinations in YaST might not work. Keys like **Alt** or **Shift** can also be occupied by the settings of the terminal.

Replacing **Alt** with **Esc**

Alt shortcuts can be executed with **Esc** instead of **Alt**. For example, **Esc-H** replaces **Alt-H**. (First press **Esc**, then press **H**.)

Backward and Forward Navigation with **Ctrl-F** and **Ctrl-B**

If the **Alt** and **Shift** combinations are occupied by the window manager or the terminal, use the combinations **Ctrl-F** (forward) and **Ctrl-B** (backward) instead.

Restriction of Function Keys

The F keys are also used for functions. Certain function keys might be occupied by the terminal and may not be available for YaST. However, the **Alt** key combinations and function keys should always be fully available on a pure text console.

4.3 YaST Command Line Options

Besides the text mode interface, YaST provides a pure command line interface. To get a list of YaST command line options, enter:

```
yast -h
```

4.3.1 Starting the Individual Modules

To save time, the individual YaST modules can be started directly. To start a module, enter:

```
yast <module_name>
```

View a list of all module names available on your system with **yast -l** or **yast --list**. Start the network module, for example, with **yast lan**.

4.3.2 Installing Packages from the Command Line

If you know a package name and the package is provided by any of your active installation repositories, you can use the command line option `-i` to install the package:

```
yast -i <package_name>
```

or

```
yast --install <package_name>
```

`package_name` can be a single short package name, for example `gvim`, which is installed with dependency checking, or the full path to an RPM package, which is installed without dependency checking.

If you need a command line based software management utility with functionality beyond what YaST provides, consider using Zypper. This utility uses the same software management library that is also the foundation for the YaST package manager. The basic usage of Zypper is covered in [Section 5.1, "Using Zypper"](#).

4.3.3 Command Line Parameters of the YaST Modules

To use YaST functionality in scripts, YaST provides command line support for individual modules. Not all modules have command line support. To display the available options of a module, enter:

```
yast <module_name> help
```

If a module does not provide command line support, the module is started in text mode and the following message appears:

```
This YaST module does not support the command line interface.
```

5 Managing Software with Command Line Tools

This chapter describes Zypper and RPM, two command line tools for managing software. For a definition of the terminology used in this context (for example, repository, patch, or update) refer to *Book “Deployment Guide”, Chapter 12 “Installing or Removing Software”, Section 12.1 “Definition of Terms”*.

5.1 Using Zypper

Zypper is a command line package manager for installing, updating and removing packages as well as for managing repositories. It is especially useful for accomplishing remote software management tasks or managing software from shell scripts.

5.1.1 General Usage

The general syntax of Zypper is:

```
tux > zypper [--global-options] command [--command-options] [arguments]
```

The components enclosed in brackets are not required. See **zypper help** for a list of general options and all commands. To get help for a specific command, type **zypper help *command***.

Zypper Commands

The simplest way to execute Zypper is to type its name, followed by a command. For example, to apply all needed patches to the system, use:

```
tux > sudo zypper patch
```

Global Options

Additionally, you can choose from one or more global options by typing them immediately before the command:

```
tux > sudo zypper --non-interactive patch
```

In the above example, the option **--non-interactive** means that the command is run without asking anything (automatically applying the default answers).

Command-Specific Options

To use options that are specific to a particular command, type them immediately after the command:

```
tux > sudo zypper patch --auto-agree-with-licenses
```

In the above example, --auto-agree-with-licenses is used to apply all needed patches to a system without you being asked to confirm any licenses. Instead, license will be accepted automatically.

Arguments

Some commands require one or more arguments. For example, when using the command **install**, you need to specify which package or which packages you want to *install*:

```
tux > sudo zypper install mplayer
```

Some options also require a single argument. The following command will list all known patterns:

```
tux > zypper search -t pattern
```

You can combine all of the above. For example, the following command will install the aspell-de and aspell-fr packages from the factory repository while being verbose:

```
tux > sudo zypper -v install --from factory aspell-de aspell-fr
```

The --from option makes sure to keep all repositories enabled (for solving any dependencies) while requesting the package from the specified repository.

Most Zypper commands have a dry-run option that does a simulation of the given command. It can be used for test purposes.

```
tux > sudo zypper remove --dry-run MozillaFirefox
```

Zypper supports the global --userdata *string* option. You can specify a string with this option, which gets written to Zypper's log files and plug-ins (such as the Btrfs plug-in). It can be used to mark and identify transactions in log files.

```
tux > sudo zypper --userdata string patch
```

5.1.2 Installing and Removing Software with Zypper

To install or remove packages, use the following commands:

```
tux > sudo zypper install package_name
tux > sudo zypper remove package_name
```



Warning: Do Not Remove Mandatory System Packages

Do not remove mandatory system packages like glibc , zypper , kernel . If they are removed, the system can become unstable or stop working altogether.

5.1.2.1 Selecting Which Packages to Install or Remove

There are various ways to address packages with the commands zypper install and zypper remove .

By Exact Package Name

```
tux > sudo zypper install MozillaFirefox
```

By Exact Package Name and Version Number

```
tux > sudo zypper install MozillaFirefox-3.5.3
```

By Repository Alias and Package Name

```
tux > sudo zypper install mozilla:MozillaFirefox
```

Where mozilla is the alias of the repository from which to install.

By Package Name Using Wild Cards

You can select all packages that have names starting or ending with a certain string. Use wild cards with care, especially when removing packages. The following command will install all packages starting with “Moz”:

```
tux > sudo zypper install 'Moz*'
```



Tip: Removing all `-debuginfo` Packages

When debugging a problem, you sometimes need to temporarily install a lot of `-debuginfo` packages which give you more information about running processes. After your debugging session finishes and you need to clean the environment, run the following:

```
tux > sudo zypper remove '*-debuginfo'
```

By Capability

For example, if you want to install a Perl module without knowing the name of the package, capabilities come in handy:

```
tux > sudo zypper install firefox
```

By Capability, Hardware Architecture, or Version

Together with a capability, you can specify a hardware architecture and a version:

- The name of the desired hardware architecture is appended to the capability after a full stop. For example, to specify the AMD64/Intel 64 architectures (which in Zypper is named `x86_64`), use:

```
tux > sudo zypper install 'firefox.x86_64'
```

- Versions must be appended to the end of the string and must be preceded by an operator: `<` (lesser than), `<=` (lesser than or equal), `=` (equal), `>=` (greater than or equal), `>` (greater than).

```
tux > sudo zypper install 'firefox>=3.5.3'
```

- You can also combine a hardware architecture and version requirement:

```
tux > sudo zypper install 'firefox.x86_64>=3.5.3'
```

By Path to the RPM file

You can also specify a local or remote path to a package:

```
tux > sudo zypper install /tmp/install/MozillaFirefox.rpm
```



```
tux > sudo zypper install http://download.opensuse.org/repositories/mozilla/
SLE_12/x86_64/MozillaFirefox-45.0.2-1.1.x86_64.rpm
```

5.1.2.2 Combining Installation and Removal of Packages

To install and remove packages simultaneously, use the `+/-` modifiers. To install `emacs` and simultaneously remove `vim`, use:

```
tux > sudo zypper install emacs -vim
```

To remove `emacs` and simultaneously install `vim`, use:

```
tux > sudo zypper remove emacs +vim
```

To prevent the package name starting with the `-` being interpreted as a command option, always use it as the second argument. If this is not possible, precede it with `--`:

```
tux > sudo zypper install -emacs +vim      # Wrong
tux > sudo zypper install vim -emacs       # Correct
tux > sudo zypper install -- -emacs +vim   # same as above
tux > sudo zypper remove emacs +vim        # same as above
```

5.1.2.3 Cleaning Up Dependencies of Removed Packages

If (together with a certain package), you automatically want to remove any packages that become unneeded after removing the specified package, use the `--clean-deps` option:

```
tux > sudo zypper rm package_name --clean-deps
```

5.1.2.4 Using Zypper in Scripts

By default, Zypper asks for a confirmation before installing or removing a selected package, or when a problem occurs. You can override this behavior using the `--non-interactive` option. This option must be given before the actual command (`install`, `remove`, and `patch`), as can be seen in the following:

```
tux > sudo zypper --non-interactive install package_name
```

This option allows the use of Zypper in scripts and cron jobs.

5.1.2.5 Installing or Downloading Source Packages

If you want to install the corresponding source package of a package, use:

```
tux > zypper source-install package_name
```

When executed as root, the default location to install source packages is /usr/src/packages/ and ~/rpmbuild when run as user. These values can be changed in your local rpm configuration.

This command will also install the build dependencies of the specified package. If you do not want this, add the switch -D. To install only the build dependencies use -d.

```
tux > sudo zypper source-install -D package_name # source package only
tux > sudo zypper source-install -d package_name # build dependencies only
```

Of course, this will only work if you have the repository with the source packages enabled in your repository list (it is added by default, but not enabled). See [Section 5.1.5, “Managing Repositories with Zypper”](#) for details on repository management.

A list of all source packages available in your repositories can be obtained with:

```
tux > zypper search -t srcpackage
```

You can also download source packages for all installed packages to a local directory. To download source packages, use:

```
tux > zypper source-download
```

The default download directory is /var/cache/zypper/source-download. You can change it using the --directory option. To only show missing or extraneous packages without downloading or deleting anything, use the --status option. To delete extraneous source packages, use the --delete option. To disable deleting, use the --no-delete option.

5.1.2.6 Installing Packages from Disabled Repositories

Normally you can only install packages from enabled repositories. The --plus-content tag option helps you specify repositories to be refreshed, temporarily enabled during the current Zypper session, and disabled after it completes.

For example, to enable repositories that may provide additional -debuginfo or -debugsource packages, use --plus-content debug. You can specify this option multiple times.

To temporarily enable such 'debug' repositories to install a specific `-debuginfo` package, use the option as follows:

```
tux > sudo zypper --plus-content debug install "debuginfo(build-id)=eb844a5c20c70a59fc693cd1061f851fb7d046f4"
```

The `build-id` string is reported by `gdb` for missing debuginfo packages.

5.1.2.7 Utilities

To verify whether all dependencies are still fulfilled and to repair missing dependencies, use:

```
tux > zypper verify
```

In addition to dependencies that must be fulfilled, some packages “recommend” other packages. These recommended packages are only installed if actually available and installable. In case recommended packages were made available after the recommending package has been installed (by adding additional packages or hardware), use the following command:

```
tux > sudo zypper install-new-recommends
```

This command is very useful after plugging in a Web cam or Wi-Fi device. It will install drivers for the device and related software, if available. Drivers and related software are only installable if certain hardware dependencies are fulfilled.

5.1.3 Updating Software with Zypper

There are three different ways to update software using Zypper: by installing patches, by installing a new version of a package or by updating the entire distribution. The latter is achieved with `zypper dist-upgrade`. Upgrading SUSE Linux Enterprise Server is discussed in *Book “Deployment Guide”, Chapter 18 “Upgrading SUSE Linux Enterprise”*.

5.1.3.1 Installing All Needed Patches

To install all officially released patches that apply to your system, run:

```
tux > sudo zypper patch
```

All patches available from repositories configured on your computer are checked for their relevance to your installation. If they are relevant (and not classified as optional or feature), they are installed immediately. Note that the official update repository is only available after registering your SUSE Linux Enterprise Server installation.

If a patch that is about to be installed includes changes that require a system reboot, you will be warned before.

To install also optional patches, use:

```
tux > sudo zypper patch --with-optional
```

To install all patches relating to a specific Bugzilla issue, use:

```
tux > sudo zypper patch --bugzilla=number
```

To install all patches relating to a specific CVE database entry, use:

```
tux > sudo zypper patch --cve=number
```

For example, to install a security patch with the CVE number CVE-2010-2713, execute:

```
tux > sudo zypper patch --cve=CVE-2010-2713
```

To install only patches which affect Zypper and the package management itself, use:

```
tux > sudo zypper patch --updatestack-only
```

5.1.3.2 Listing Patches

To find out whether patches are available, Zypper allows viewing the following information:

Number of Needed Patches

To list the number of needed patches (patches that apply to your system but are not yet installed), use **patch-check**:

```
tux > zypper patch-check
Loading repository data...
Reading installed packages...
5 patches needed (1 security patch)
```

This command can be combined with the `--updatestack-only` option to list only the patches which affect Zypper and the package management itself.

List of Needed Patches

To list all needed patches (patches that apply to your system but are not yet installed), use `list-patches`:

```
tux > zypper list-patches
Loading repository data...
Reading installed packages...

Repository | Name | Version | Category | Status | Summary
-----+-----+-----+-----+-----+-----
SLES12-Updates | SUSE-2014-8 | 1 | security | needed | openssl: Update for OpenSSL
```

List of All Patches

To list all patches available for SUSE Linux Enterprise Server, regardless of whether they are already installed or apply to your installation, use `zypper patches`.

It is also possible to list and install patches relevant to specific issues. To list specific patches, use the `zypper list-patches` command with the following options:

By Bugzilla Issues

To list all needed patches that relate to Bugzilla issues, use the option `--bugzilla`.

To list patches for a specific bug, you can also specify a bug number: `--bugzilla=number`.

To search for patches relating to multiple Bugzilla issues, add commas between the bug numbers, for example:

```
tux > zypper list-patches --bugzilla=972197,956917
```

By CVE Number

To list all needed patches that relate to an entry in the CVE database (Common Vulnerabilities and Exposures), use the option `--cve`.

To list patches for a specific CVE database entry, you can also specify a CVE number: `--cve=number`. To search for patches relating to multiple CVE database entries, add commas between the CVE numbers, for example:

```
tux > zypper list-patches --bugzilla=CVE-2016-2315,CVE-2016-2324
```

To list all patches regardless of whether they are needed, use the option `--all` additionally. For example, to list all patches with a CVE number assigned, use:

```
tux > zypper list-patches --all --cve
Issue | No.          | Patch              | Category   | Severity   | Status
-----+-----+-----+-----+-----+-----
cve    | CVE-2015-0287 | SUSE-SLE-Module.. | recommended | moderate   | needed
cve    | CVE-2014-3566 | SUSE-SLE-SERVER.. | recommended | moderate   | not needed
[...]
```

5.1.3.3 Installing New Package Versions

If a repository contains only new packages, but does not provide patches, **zypper patch** does not show any effect. To update all installed packages with newer available versions (while maintaining system integrity), use:

```
tux > sudo zypper update
```

To update individual packages, specify the package with either the update or install command:

```
tux > sudo zypper update package_name
tux > sudo zypper install package_name
```

A list of all new installable packages can be obtained with the command:

```
tux > zypper list-updates
```

Note that this command only lists packages that match the following criteria:

- has the same vendor like the already installed package,
- is provided by repositories with at least the same priority than the already installed package,
- is installable (all dependencies are satisfied).

A list of *all* new available packages (regardless whether installable or not) can be obtained with:

```
tux > sudo zypper list-updates --all
```

To find out why a new package cannot be installed, use the **zypper install** or **zypper update** command as described above.

5.1.3.4 Identifying Orphaned Packages

Whenever you remove a repository from Zypper or upgrade your system, some packages can get in an “orphaned” state. These *orphaned* packages belong to no active repository anymore. The following command gives you a list of these:

```
tux > sudo zypper packages --orphaned
```

With this list, you can decide if a package is still needed or can be removed safely.

5.1.4 Identifying Processes and Services Using Deleted Files

When patching, updating or removing packages, there may be running processes on the system which continue to use files having been deleted by the update or removal. Use **zypper ps** to list processes using deleted files. In case the process belongs to a known service, the service name is listed, making it easy to restart the service. By default **zypper ps** shows a table:

PID	PPID	UID	User	Command	Service	Files
814	1	481	avahi	avahi-daemon	avahi-daemon	/lib64/ld-2.19.s->
						/lib64/libdl-2.1->
						/lib64/libpthrea->
						/lib64/libc-2.19->
[...]						

PID: ID of the process

PPID: ID of the parent process

UID: ID of the user running the process

Login: Login name of the user running the process

Command: Command used to execute the process

Service: Service name (only if command is associated with a system service)

Files: The list of the deleted files

The output format of **zypper ps** can be controlled as follows:

zypper ps -s

Create a short table not showing the deleted files.

PID	PPID	UID	User	Command	Service
-----	------	-----	------	---------	---------

```

-----+-----+-----+-----+-----+-----
814   | 1     | 481  | avahi  | avahi-daemon | avahi-daemon
817   | 1     | 0     | root   | irqbalance   | irqbalance
1567  | 1     | 0     | root   | sshd         | sshd
1761  | 1     | 0     | root   | master       | postfix
1764  | 1761  | 51    | postfix| pickup       | postfix
1765  | 1761  | 51    | postfix| qmgr         | postfix
2031  | 2027  | 1000  | tux    | bash         |

```

zypper ps -ss

Show only processes associated with a system service.

```

PID    | PPID | UID  | User   | Command       | Service
-----+-----+-----+-----+-----+-----
814    | 1    | 481  | avahi  | avahi-daemon  | avahi-daemon
817    | 1    | 0     | root   | irqbalance    | irqbalance
1567   | 1    | 0     | root   | sshd          | sshd
1761   | 1    | 0     | root   | master        | postfix
1764   | 1761 | 51    | postfix| pickup        | postfix
1765   | 1761 | 51    | postfix| qmgr          | postfix

```

zypper ps -sss

Only show system services using deleted files.

```

avahi-daemon
irqbalance
postfix
sshd

```

zypper ps --print "systemctl status %s"

Show the commands to retrieve status information for services which might need a restart.

```

systemctl status avahi-daemon
systemctl status irqbalance
systemctl status postfix
systemctl status sshd

```

For more information about service handling refer to [Chapter 13, The systemd Daemon](#).

5.1.5 Managing Repositories with Zypper

All installation or patch commands of Zypper rely on a list of known repositories. To list all repositories known to the system, use the command:

```
tux > zypper repos
```

The result will look similar to the following output:

EXAMPLE 5.1: ZYPPER—LIST OF KNOWN REPOSITORIES

#	Alias	Name	Enabled	Refresh
1	SLEHA-12-GE0	SLEHA-12-GE0	Yes	No
2	SLEHA-12	SLEHA-12	Yes	No
3	SLES12	SLES12	Yes	No

When specifying repositories in various commands, an alias, URI or repository number from the **zypper repos** command output can be used. A repository alias is a short version of the repository name for use in repository handling commands. Note that the repository numbers can change after modifying the list of repositories. The alias will never change by itself.

By default, details such as the URI or the priority of the repository are not displayed. Use the following command to list all details:

```
tux > zypper repos -d
```

5.1.5.1 Adding Repositories

To add a repository, run

```
tux > sudo zypper addrepo URI alias
```

URI can either be an Internet repository, a network resource, a directory or a CD or DVD (see http://en.opensuse.org/openSUSE:Libzypp_URIs for details). The alias is a shorthand and unique identifier of the repository. You can freely choose it, with the only exception that it needs to be unique. Zypper will issue a warning if you specify an alias that is already in use.

5.1.5.2 Removing Repositories

If you want to remove a repository from the list, use the command **zypper removerepo** together with the alias or number of the repository you want to delete. For example, to remove the repository SLEHA-12-GE0 from *Example 5.1, “Zypper—List of Known Repositories”*, use one of the following commands:

```
tux > sudo zypper removerepo 1
tux > sudo zypper removerepo "SLEHA-12-GE0"
```

5.1.5.3 Modifying Repositories

Enable or disable repositories with **zypper modifyrepo**. You can also alter the repository's properties (such as refreshing behavior, name or priority) with this command. The following command will enable the repository named updates, turn on auto-refresh and set its priority to 20:

```
tux > sudo zypper modifyrepo -er -p 20 'updates'
```

Modifying repositories is not limited to a single repository—you can also operate on groups:

-a: all repositories

-l: local repositories

-r: remote repositories

-m TYPE: repositories of a certain type (where TYPE can be one of the following: http, https, ftp, cd, dvd, dir, file, cifs, smb, nfs, hd, iso)

To rename a repository alias, use the **renamerepo** command. The following example changes the alias from Mozilla Firefox to firefox:

```
tux > sudo zypper renamerepo 'Mozilla Firefox' firefox
```

5.1.6 Querying Repositories and Packages with Zypper

Zypper offers various methods to query repositories or packages. To get lists of all products, patterns, packages or patches available, use the following commands:

```
tux > zypper products
```

```
tux > zypper patterns
tux > zypper packages
tux > zypper patches
```

To query all repositories for certain packages, use search. It works on package names, or, optionally, on package summaries and descriptions. String wrapped in / are interpreted as regular expressions. By default, the search is not case-sensitive.

Simple search for a package name containing fire

```
tux > zypper search "fire"
```

Simple search for the exact package MozillaFirefox

```
tux > zypper search --match-exact "MozillaFirefox"
```

Also search in package descriptions and summaries

```
tux > zypper search -d fire
```

Only display packages not already installed

```
tux > zypper search -u fire
```

Display packages containing the string fir not followed by e

```
tux > zypper se "/fir[^e]/"
```

To search for packages which provide a special capability, use the command what-provides. For example, if you want to know which package provides the Perl module SVN::Core, use the following command:

```
tux > zypper what-provides 'perl(SVN::Core)'
```

To query single packages, use info with an exact package name as an argument. It displays detailed information about a package. To also show what is required/recommended by the package, use the options --requires and --recommends:

```
tux > zypper info --requires MozillaFirefox
```

The `what-provides package` is similar to `rpm -q --whatprovides package`, but RPM is only able to query the RPM database (that is the database of all installed packages). Zypper, on the other hand, will tell you about providers of the capability from any repository, not only those that are installed.

5.1.7 Configuring Zypper

Zypper now comes with a configuration file, allowing you to permanently change Zypper's behavior (either system-wide or user-specific). For system-wide changes, edit `/etc/zypp/zypper.conf`. For user-specific changes, edit `~/.zypper.conf`. If `~/.zypper.conf` does not yet exist, you can use `/etc/zypp/zypper.conf` as a template: copy it to `~/.zypper.conf` and adjust it to your liking. Refer to the comments in the file for help about the available options.

5.1.8 Troubleshooting

In case you have problems to access packages from configured repositories (for example, Zypper cannot find a certain package though you know that it exists in one the repositories), it can help to refresh the repositories with:

```
tux > sudo zypper refresh
```

If that does not help, try



```
tux > sudo zypper refresh -fdb
```

This forces a complete refresh and rebuild of the database, including a forced download of raw metadata.

5.1.9 Zypper Rollback Feature on Btrfs File System

If the Btrfs file system is used on the root partition and **snapper** is installed, Zypper automatically calls **snapper** (via script installed by **snapper**) when committing changes to the file system to create appropriate file system snapshots. These snapshots can be used for reverting any changes made by Zypper. See *Chapter 6, System Recovery and Snapshot Management with Snapper* for more information.

5.1.10 For More Information

For more information on managing software from the command line, enter `zypper help`, `zypper help command` or refer to the `zypper(8)` man page. For a complete and detailed command reference, including cheat sheets with the most important commands, and information on how to use Zypper in scripts and applications, refer to http://en.opensuse.org/SDB:Zypper_usage . A list of software changes for the latest SUSE Linux Enterprise Server version can be found at http://en.opensuse.org/openSUSE:Zypper_versions .

5.2 RPM—the Package Manager

RPM (RPM Package Manager) is used for managing software packages. Its main commands are `rpm` and `rpmbuild`. The powerful RPM database can be queried by the users, system administrators and package builders for detailed information about the installed software.

Essentially, `rpm` has five modes: installing, uninstalling (or updating) software packages, rebuilding the RPM database, querying RPM bases or individual RPM archives, integrity checking of packages and signing packages. `rpmbuild` can be used to build installable packages from pristine sources.

Installable RPM archives are packed in a special binary format. These archives consist of the program files to install and certain meta information used during the installation by `rpm` to configure the software package or stored in the RPM database for documentation purposes. RPM archives normally have the extension `.rpm`.



Tip: Software Development Packages

For several packages, the components needed for software development (libraries, headers, include files, etc.) have been put into separate packages. These development packages are only needed if you want to compile software yourself (for example, the most recent GNOME packages). They can be identified by the name extension `-devel`, such as the packages `alsa-devel` and `gimp-devel`.

5.2.1 Verifying Package Authenticity

RPM packages have a GPG signature. To verify the signature of an RPM package, use the command `rpm --checksig package-1.2.3.rpm` to determine whether the package originates from SUSE or from another trustworthy facility. This is especially recommended for update packages from the Internet.

While fixing issues in the operating system, you might need to install a Problem Temporary Fix (PTF) into a production system. The packages provided by SUSE are signed against a special PTF key. However, in contrast to SUSE Linux Enterprise 11, this key is not imported by default on SUSE Linux Enterprise 12 systems. To manually import the key, use the following command:

```
rpm --import /usr/share/doc/packages/suse-build-key/suse_ptf_key.asc
```

After importing the key, you can install PTF packages on your system.

5.2.2 Managing Packages: Install, Update, and Uninstall

Normally, the installation of an RPM archive is quite simple: `rpm -i package.rpm`. With this command the package is installed, but only if its dependencies are fulfilled and if there are no conflicts with other packages. With an error message, `rpm` requests those packages that need to be installed to meet dependency requirements. In the background, the RPM database ensures that no conflicts arise—a specific file can only belong to one package. By choosing different options, you can force `rpm` to ignore these defaults, but this is only for experts. Otherwise, you risk compromising the integrity of the system and possibly jeopardize the ability to update the system.

The options `-U` or `--upgrade` and `-F` or `--freshen` can be used to update a package (for example, `rpm -F package.rpm`). This command removes the files of the old version and immediately installs the new files. The difference between the two versions is that `-U` installs packages that previously did not exist in the system, but `-F` merely updates previously installed packages. When updating, `rpm` updates configuration files carefully using the following strategy:

- If a configuration file was not changed by the system administrator, `rpm` installs the new version of the appropriate file. No action by the system administrator is required.
- If a configuration file was changed by the system administrator before the update, `rpm` saves the changed file with the extension `.rpmorig` or `.rpmsave` (backup file) and installs the version from the new package (but only if the originally installed file and the newer

version are different). If this is the case, compare the backup file (.rpmorig or .rpmsave) with the newly installed file and make your changes again in the new file. Afterwards, be sure to delete all .rpmorig and .rpmsave files to avoid problems with future updates.

- .rpmnew files appear if the configuration file already exists *and* if the noreplace label was specified in the .spec file.

Following an update, .rpmsave and .rpmnew files should be removed after comparing them, so they do not obstruct future updates. The .rpmorig extension is assigned if the file has not previously been recognized by the RPM database.

Otherwise, .rpmsave is used. In other words, .rpmorig results from updating from a foreign format to RPM. .rpmsave results from updating from an older RPM to a newer RPM. .rpmnew does not disclose any information to whether the system administrator has made any changes to the configuration file. A list of these files is available in /var/adm/rpmconfigcheck. Some configuration files (like /etc/httpd/httpd.conf) are not overwritten to allow continued operation.

The -U switch is *not* just an equivalent to uninstalling with the -e option and installing with the -i option. Use -U whenever possible.

To remove a package, enter **rpm -e package**. This command only deletes the package if there are no unresolved dependencies. It is theoretically impossible to delete Tcl/Tk, for example, as long as another application requires it. Even in this case, RPM calls for assistance from the database. If such a deletion is, for whatever reason, impossible (even if *no* additional dependencies exist), it may be helpful to rebuild the RPM database using the option --rebuilddb.

5.2.3 Delta RPM Packages

Delta RPM packages contain the difference between an old and a new version of an RPM package. Applying a delta RPM onto an old RPM results in a completely new RPM. It is not necessary to have a copy of the old RPM because a delta RPM can also work with an installed RPM. The delta RPM packages are even smaller in size than patch RPMs, which is an advantage when transferring update packages over the Internet. The drawback is that update operations with delta RPMs involved consume considerably more CPU cycles than plain or patch RPMs.

The `makedeltarpm` and `applydelta` binaries are part of the delta RPM suite (package `deltarpm`) and help you create and apply delta RPM packages. With the following commands, you can create a delta RPM called `new.delta.rpm`. The following command assumes that `old.rpm` and `new.rpm` are present:

```
makedeltarpm old.rpm new.rpm new.delta.rpm
```

Using `applydeltarpm`, you can reconstruct the new RPM from the file system if the old package is already installed:

```
applydeltarpm new.delta.rpm new.rpm
```

To derive it from the old RPM without accessing the file system, use the `-r` option:

```
applydeltarpm -r old.rpm new.delta.rpm new.rpm
```

See </usr/share/doc/packages/deltarpm/README> for technical details.

5.2.4 RPM Queries

With the `-q` option `rpm` initiates queries, making it possible to inspect an RPM archive (by adding the option `-p`) and to query the RPM database of installed packages. Several switches are available to specify the type of information required. See [Table 5.1, “The Most Important RPM Query Options”](#).

TABLE 5.1: THE MOST IMPORTANT RPM QUERY OPTIONS

<code>-i</code>	Package information
<code>-l</code>	File list
<code>-f FILE</code>	Query the package that contains the file <code>FILE</code> (the full path must be specified with <code>FILE</code>)
<code>-s</code>	File list with status information (implies <code>-l</code>)
<code>-d</code>	List only documentation files (implies <code>-l</code>)
<code>-c</code>	List only configuration files (implies <code>-l</code>)

<u>--dump</u>	File list with complete details (to be used with <u>-l</u> , <u>-c</u> , or <u>-d</u>)
<u>--provides</u>	List features of the package that another package can request with <u>--requires</u>
<u>--requires</u> , <u>-R</u>	Capabilities the package requires
<u>--scripts</u>	Installation scripts (preinstall, postinstall, uninstall)

For example, the command `rpm -q -i wget` displays the information shown in *Example 5.2*, “`rpm -q -i wget`”.

EXAMPLE 5.2: `rpm -q -i wget`

```
Name      : wget                               Relocations: (not relocatable)
Version   : 1.11.4                             Vendor: openSUSE
Release   : 1.70                               Build Date: Sat 01 Aug 2009 09:49:48
CEST
Install Date: Thu 06 Aug 2009 14:53:24 CEST      Build Host: build18
Group     : Productivity/Networking/Web/Utilities Source RPM:
  wget-1.11.4-1.70.src.rpm
Size      : 1525431                             License: GPL v3 or later
Signature : RSA/8, Sat 01 Aug 2009 09:50:04 CEST, Key ID b88b2fd43dbdc284
Packager  : http://bugs.opensuse.org
URL       : http://www.gnu.org/software/wget/
Summary   : A Tool for Mirroring FTP and HTTP Servers
Description :
Wget enables you to retrieve WWW documents or FTP files from a server.
This can be done in script files or via the command line.
[...]
```

The option -f only works if you specify the complete file name with its full path. Provide as many file names as desired. For example, the following command

```
rpm -q -f /bin/rpm /usr/bin/wget
```

results in:

```
rpm-4.8.0-4.3.x86_64
wget-1.11.4-11.18.x86_64
```

If only part of the file name is known, use a shell script as shown in *Example 5.3, “Script to Search for Packages”*. Pass the partial file name to the script shown as a parameter when running it.

EXAMPLE 5.3: SCRIPT TO SEARCH FOR PACKAGES

```
#!/bin/sh
for i in $(rpm -q -a -l | grep $1); do
    echo "\"$i\" is in package:"
    rpm -q -f $i
    echo ""
done
```

The command `rpm -q --changelog package` displays a detailed list of change information about a specific package, sorted by date.

With the installed RPM database, verification checks can be made. Initiate these with `-V`, or `--verify`. With this option, `rpm` shows all files in a package that have been changed since installation. `rpm` uses eight character symbols to give some hints about the following changes:

TABLE 5.2: RPM VERIFY OPTIONS

<u>5</u>	MD5 check sum
<u>S</u>	File size
<u>L</u>	Symbolic link
<u>T</u>	Modification time
<u>D</u>	Major and minor device numbers
<u>U</u>	Owner
<u>G</u>	Group
<u>M</u>	Mode (permissions and file type)

In the case of configuration files, the letter c is printed. For example, for changes to `/etc/wgetrc` (`wget` package):

```
rpm -V wget
S.5....T c /etc/wgetrc
```

The files of the RPM database are placed in `/var/lib/rpm`. If the partition `/usr` has a size of 1 GB, this database can occupy nearly 30 MB, especially after a complete update. If the database is much larger than expected, it is useful to rebuild the database with the option `--rebuilddb`. Before doing this, make a backup of the old database. The `cron` script `cron.daily` makes daily copies of the database (packed with gzip) and stores them in `/var/adm/backup/rpmdb`. The number of copies is controlled by the variable `MAX_RPMDDB_BACKUPS` (default: 5) in `/etc/sysconfig/backup`. The size of a single backup is approximately 1 MB for 1 GB in `/usr`.

5.2.5 Installing and Compiling Source Packages

All source packages carry a `.src.rpm` extension (source RPM).



Note: Installed Source Packages

Source packages can be copied from the installation medium to the hard disk and unpacked with YaST. They are not, however, marked as installed (`[i]`) in the package manager. This is because the source packages are not entered in the RPM database. Only *installed* operating system software is listed in the RPM database. When you “install” a source package, only the source code is added to the system.

The following directories must be available for `rpm` and `rpmbuild` in `/usr/src/packages` (unless you specified custom settings in a file like `/etc/rpmrc`):

SOURCES

for the original sources (`.tar.bz2` or `.tar.gz` files, etc.) and for distribution-specific adjustments (mostly `.diff` or `.patch` files)

SPECS

for the `.spec` files, similar to a meta Makefile, which control the *build* process

BUILD

all the sources are unpacked, patched and compiled in this directory

RPMS

where the completed binary packages are stored

SRPMS

here are the source RPMs

When you install a source package with YaST, all the necessary components are installed in /usr/src/packages: the sources and the adjustments in SOURCES and the relevant .spec file in SPECS.



Warning: System Integrity

Do not experiment with system components (glibc, rpm, etc.), because this endangers the stability of your system.

The following example uses the wget.src.rpm package. After installing the source package, you should have files similar to those in the following list:

```
/usr/src/packages/SOURCES/wget-1.11.4.tar.bz2
/usr/src/packages/SOURCES/wgetrc.patch
/usr/src/packages/SPECS/wget.spec
```

rpmbuild -bX /usr/src/packages/SPECS/wget.spec starts the compilation. X is a wild card for various stages of the build process (see the output of --help or the RPM documentation for details). The following is merely a brief explanation:

-bp

Prepare sources in /usr/src/packages/BUILD: unpack and patch.

-bc

Do the same as -bp, but with additional compilation.

-bi

Do the same as -bp, but with additional installation of the built software. Caution: if the package does not support the BuildRoot feature, you might overwrite configuration files.

-bb

Do the same as -bi, but with the additional creation of the binary package. If the compile was successful, the binary should be in /usr/src/packages/RPMS.

-ba

Do the same as -bb, but with the additional creation of the source RPM. If the compilation was successful, the binary should be in /usr/src/packages/SRPMS.

--short-circuit

Skip some steps.

The binary RPM created can now be installed with `rpm -i` or, preferably, with `rpm -U`. Installation with `rpm` makes it appear in the RPM database.

Keep in mind, the `BuildRoot` directive in the spec file was deprecated since SLE12 and above. If you still need this feature, use the `--buildroot` option as a workaround. For a more detailed background, see the support database at <https://www.suse.com/support/kb/doc?id=7017104>.

5.2.6 Compiling RPM Packages with build

The danger with many packages is that unwanted files are added to the running system during the build process. To prevent this use `build`, which creates a defined environment in which the package is built. To establish this chroot environment, the `build` script must be provided with a complete package tree. This tree can be made available on the hard disk, via NFS, or from DVD. Set the position with `build --rpms directory`. Unlike `rpm`, the `build` command looks for the `.spec` file in the source directory. To build `wget` (like in the above example) with the DVD mounted in the system under `/media/dvd`, use the following commands as `root`:

```
cd /usr/src/packages/SOURCES/  
mv ../SPECS/wget.spec .  
build --rpms /media/dvd/suse/ wget.spec
```

Subsequently, a minimum environment is established at `/var/tmp/build-root`. The package is built in this environment. Upon completion, the resulting packages are located in `/var/tmp/build-root/usr/src/packages/RPMS`.

The `build` script offers several additional options. For example, cause the script to prefer your own RPMs, omit the initialization of the build environment or limit the `rpm` command to one of the above-mentioned stages. Access additional information with `build --help` and by reading the `build` man page.

5.2.7 Tools for RPM Archives and the RPM Database

Midnight Commander (`mc`) can display the contents of RPM archives and copy parts of them. It represents archives as virtual file systems, offering all usual menu options of Midnight Commander. Display the `HEADER` with `F3`. View the archive structure with the cursor keys and `Enter`. Copy archive components with `F5`.

A full-featured package manager is available as a YaST module. For details, see *Book “Deployment Guide”, Chapter 12 “Installing or Removing Software”*.

6 System Recovery and Snapshot Management with Snapper

Being able to do file system snapshots providing the ability to do rollbacks on Linux is a feature that was often requested in the past. Snapper, with the Btrfs file system or thin-provisioned LVM volumes now fills that gap.

Btrfs, a new copy-on-write file system for Linux, supports file system snapshots (a copy of the state of a subvolume at a certain point of time) of subvolumes (one or more separately mountable file systems within each physical partition). Snapshots are also supported on thin-provisioned LVM volumes formatted with XFS, Ext4 or Ext3. Snapper lets you create and manage these snapshots. It comes with a command line and a YaST interface. Starting with SUSE Linux Enterprise Server 12 it is also possible to boot from Btrfs snapshots—see [Section 6.3, “System Rollback by Booting from Snapshots”](#) for more information.

Using Snapper you can perform the following tasks:

- Undo system changes made by zypper and YaST. See [Section 6.2, “Using Snapper to Undo Changes”](#) for details.
- Restore files from previous snapshots. See [Section 6.2.2, “Using Snapper to Restore Files”](#) for details.
- Do a system rollback by booting from a snapshot. See [Section 6.3, “System Rollback by Booting from Snapshots”](#) for details.
- Manually create snapshots on the fly and manage existing snapshots. See [Section 6.5, “Manually Creating and Managing Snapshots”](#) for details.

6.1 Default Setup

Snapper on SUSE Linux Enterprise Server is set up to serve as an “undo and recovery tool” for system changes. By default, the root partition (/) of SUSE Linux Enterprise Server is formatted with Btrfs. Taking snapshots is automatically enabled if the root partition (/) is big enough (approximately more than 16 GB). Taking snapshots on partitions other than / is not enabled by default.



Tip: Enabling Snapper in the Installed System

If you have disabled Snapper during the installation, you can enable it at any time later. To do so, create a default Snapper configuration for the root file system by running

```
sudo snapper -c root create-config /
```

Afterward enable the different snapshot types as described in [Section 6.1.3.1, “Disabling/Enabling Snapshots”](#).

Keep in mind that snapshots require a Btrfs root file system with subvolumes set up as proposed by the installer and a partition size of at least 16 GB.

When a snapshot is created, both the snapshot and the original point to the same blocks in the file system. So, initially a snapshot does not occupy additional disk space. If data in the original file system is modified, changed data blocks are copied while the old data blocks are kept for the snapshot. Therefore, a snapshot occupies the same amount of space as the data modified. So, over time, the amount of space a snapshot allocates, constantly grows. As a consequence, deleting files from a Btrfs file system containing snapshots may *not* free disk space!



Note: Snapshot Location

Snapshots always reside on the same partition or subvolume on which the snapshot has been taken. It is not possible to store snapshots on a different partition or subvolume.

As a result, partitions containing snapshots need to be larger than “normal” partitions. The exact amount strongly depends on the number of snapshots you keep and the amount of data modifications. As a rule of thumb you should consider using twice the size than you normally would. To prevent disks from running out of space, old snapshots are automatically cleaned up. Refer to [Section 6.1.3.4, “Controlling Snapshot Archiving”](#) for details.

6.1.1 Types of Snapshots

Although snapshots themselves do not differ in a technical sense, we distinguish between three types of snapshots, based on the occasion on which they were taken:

Timeline Snapshots

A single snapshot is created every hour. Old snapshots are automatically deleted. By default, the first snapshot of the last ten days, months, and years are kept. Timeline snapshots are disabled by default.

Installation Snapshots

Whenever one or more packages are installed with YaST or Zypper, a pair of snapshots is created: one before the installation starts (“Pre”) and another one after the installation has finished (“Post”). In case an important system component such as the kernel has been installed, the snapshot pair is marked as important (`important=yes`). Old snapshots are automatically deleted. By default the last ten important snapshots and the last ten “regular” (including administration snapshots) snapshots are kept. Installation snapshots are enabled by default.

Administration Snapshots

Whenever you administrate the system with YaST, a pair of snapshots is created: one when a YaST module is started (“Pre”) and another when the module is closed (“Post”). Old snapshots are automatically deleted. By default the last ten important snapshots and the last ten “regular” snapshots (including installation snapshots) are kept. Administration snapshots are enabled by default.

6.1.2 Directories That Are Excluded from Snapshots

Some directories need to be excluded from snapshots for different reasons. The following list shows all directories that are excluded:

/boot/grub2/i386-pc, /boot/grub2/x86_64-efi, /boot/grub2/powerpc-ieee1275, /boot/grub2/s390x-emu

A rollback of the boot loader configuration is not supported. The directories listed above are architecture-specific. The first two directories are present on AMD64/Intel 64 machines, the latter two on IBM POWER and on IBM z Systems, respectively.

/home

If /home does not reside on a separate partition, it is excluded to avoid data loss on rollbacks.

/opt, /var/opt

Third-party products usually get installed to /opt. It is excluded to avoid uninstalling these applications on rollbacks.

/srv

Contains data for Web and FTP servers. It is excluded to avoid data loss on rollbacks.

/tmp, /var/tmp, /var/cache, /var/crash

All directories containing temporary files and caches are excluded from snapshots.

/usr/local

This directory is used when manually installing software. It is excluded to avoid uninstalling these installations on rollbacks.

/var/lib/libvirt/images

The default location for virtual machine images managed with libvirt. Excluded to ensure virtual machine images are not replaced with older versions during a rollback. By default, this subvolume is created with the option no copy on write.

/var/lib/mailman, /var/spool

Directories containing mails or mail queues are excluded to avoid a loss of mails after a rollback.

/var/lib/named

Contains zone data for the DNS server. Excluded from snapshots to ensure a name server can operate after a rollback.

/var/lib/mariadb, /var/lib/mysql, /var/lib/pgqsl

These directories contain database data. By default, these subvolumes are created with the option no copy on write.

/var/log

Log file location. Excluded from snapshots to allow log file analysis after the rollback of a broken system.

6.1.3 Customizing the Setup

SUSE Linux Enterprise Server comes with a reasonable default setup, which should be sufficient for most use cases. However, all aspects of taking automatic snapshots and snapshot keeping can be configured according to your needs.

6.1.3.1 Disabling/Enabling Snapshots

Each of the three snapshot types (timeline, installation, administration) can be enabled or disabled independently.

Disabling/Enabling Timeline Snapshots

Enabling. `snapper-c root set-config "TIMELINE_CREATE=yes"`

Disabling. `snapper -c root set-config "TIMELINE_CREATE=no"`

Timeline snapshots are enabled by default, except for the root partition.

Disabling/Enabling Installation Snapshots

Enabling: Install the package `snapper-zypp-plugin`

Disabling: Uninstall the package `snapper-zypp-plugin`

Installation snapshots are enabled by default.

Disabling/Enabling Administration Snapshots

Enabling: Set `USE_SNAPPER` to `yes` in `/etc/sysconfig/yast2`.

Disabling: Set `USE_SNAPPER` to `no` in `/etc/sysconfig/yast2`.

Administration snapshots are enabled by default.

6.1.3.2 Controlling Installation Snapshots

Taking snapshot pairs upon installing packages with YaST or Zypper is handled by the `snapper-zypp-plugin`. An XML configuration file, `/etc/snapper/zypp-plugin.conf` defines, when to make snapshots. By default the file looks like the following:

```
1 <?xml version="1.0" encoding="utf-8"?>
2 <snapper-zypp-plugin-conf>
3   <solvables>
4     <solvable match="w" ❶ important="true" ❷>kernel-* ❸</solvable>
5     <solvable match="w" important="true">dracut</solvable>
```

```

6 <solvable match="w" important="true">glibc</solvable>
7 <solvable match="w" important="true">systemd*</solvable>
8 <solvable match="w" important="true">udev</solvable>
9 <solvable match="w">*</solvable> ④
10 </solvables>
11 </snapper-zypp-plugin-conf>

```

- ① The `match` attribute defines whether the pattern is a Unix shell-style wild card (w) or a Python regular expression (re).
- ② If the given pattern matches and the corresponding package is marked as important (for example Kernel packages), the snapshot will also be marked as important.
- ③ Pattern to match a package name. Based on the setting of the `match` attribute, special characters are either interpreted as shell wild cards or regular expressions. This pattern matches all package names starting with `kernel-`.
- ④ This line unconditionally matches all packages.

With this configuration snapshot, pairs are made whenever a package is installed (line 9). When Kernel, dracut, glibc, systemd, or udev packages marked as important are installed, the snapshot pair will also be marked as important (lines 4 to 8). All rules are evaluated.

To disable a rule, either delete it or deactivate it using XML comments. To prevent the system from making snapshot pairs for every package installation for example, comment line 9:

```

1 <?xml version="1.0" encoding="utf-8"?>
2 <snapper-zypp-plugin-conf>
3 <solvables>
4 <solvable match="w" important="true">kernel-*</solvable>
5 <solvable match="w" important="true">dracut</solvable>
6 <solvable match="w" important="true">glibc</solvable>
7 <solvable match="w" important="true">systemd*</solvable>
8 <solvable match="w" important="true">udev</solvable>
9 <!-- <solvable match="w">*</solvable> -->
10 </solvables>
11 </snapper-zypp-plugin-conf>

```

6.1.3.3 Creating and Mounting New Subvolumes

Creating a new subvolume underneath the `/`-hierarchy and permanently mounting it is supported. However, you need to make sure not to create it inside a snapshot, since you would not be able to delete snapshots anymore after a rollback.

SUSE Linux Enterprise Server is configured with the `/@/` subvolume which serves as an independent root for permanent subvolumes such as `/opt`, `/srv`, `/home` and others. Any new subvolumes you create and permanently mount need to be created in this initial root file system. To do so, run the following commands. In this example, a new subvolume `/usr/important` is created from `/dev/sda2`.

```
mount /dev/sda2 -o subvol=@ /mnt
btrfs subvolume create /mnt/usr/important
umount /mnt
```

The corresponding entry in `/etc/fstab` needs to look like the following:

```
/dev/sda2 /usr/important btrfs subvol=@/usr/important 0 0
```

6.1.3.4 Controlling Snapshot Archiving

Snapshots occupy disk space. To prevent disks from running out of space and thus causing system outages, old snapshots are automatically deleted. By default, up to ten important installation and administration snapshots and up to ten regular installation and administration snapshots are kept. If these snapshots occupy more than 50% of the root file system size, additional snapshots will be deleted. A minimum of four important and two regular snapshots are always kept.

Refer to [Section 6.4.1, “Managing Existing Configurations”](#) for instructions on how to change these values.

6.1.3.5 Using Snapper on Thin-Provisioned LVM Volumes

Apart from snapshots on `Btrfs` file systems, Snapper also supports taking snapshots on thin-provisioned LVM volumes (snapshots on regular LVM volumes are *not* supported) formatted with XFS, Ext4 or Ext3. For more information and setup instructions on LVM volumes, refer to *Book “Deployment Guide”, Chapter 11 “Advanced Disk Setup”, Section 11.2 “LVM Configuration”*. To use Snapper on a thin-provisioned LVM volume you need to create a Snapper configuration for it. On LVM it is required to specify the file system with `--fstype=lvm(FILESYSTEM)`. `ext3`, `ext4` or `xfs` are valid values for `FILESYSTEM`. Example:

```
snapper -c lvm create-config --fstype="lvm(xfs)" /thin_lvm
```

You can adjust this configuration according to your needs as described in [Section 6.4.1, “Managing Existing Configurations”](#).

6.2 Using Snapper to Undo Changes

Snapper on SUSE Linux Enterprise Server is preconfigured to serve as a tool that lets you undo changes made by **zypper** and YaST. For this purpose, Snapper is configured to create a pair of snapshots before and after each run of **zypper** and YaST. Snapper also lets you restore system files that have been accidentally deleted or modified. Timeline snapshots for the root partition need to be enabled for this purpose—see [Section 6.1.3.1, “Disabling/Enabling Snapshots”](#) for details. By default, automatic snapshots as described above are configured for the root partition and its subvolumes. To make snapshots available for other partitions such as `/home` for example, you can create custom configurations.



Important: Undoing Changes Compared to Rollback

When working with snapshots to restore data, it is important to know that there are two fundamentally different scenarios Snapper can handle:

Undoing Changes

When undoing changes as described in the following, two snapshots are being compared and the changes between these two snapshots are made undone. Using this method also allows to explicitly select the files that should be restored.

Rollback

When doing rollbacks as described in [Section 6.3, “System Rollback by Booting from Snapshots”](#), the system is reset to the state at which the snapshot was taken.

When undoing changes, it is also possible to compare a snapshot against the current system. When restoring *all* files from such a comparison, this will have the same result as doing a rollback. However, using the method described in [Section 6.3, “System Rollback by Booting from Snapshots”](#) for rollbacks should be preferred, since it is faster and allows you to review the system before doing the rollback.



Warning: Data Consistency

There is no mechanism to ensure data consistency when creating a snapshot. Whenever a file (for example, a database) is written at the same time as the snapshot is being created, it will result in a broken or partly written file. Restoring such a file will cause problems. Furthermore, some system files such as `/etc/mtab` must never be restored. Therefore it is strongly recommended to *always* closely review the list of changed files and their diffs. Only restore files that really belong to the action you want to revert.

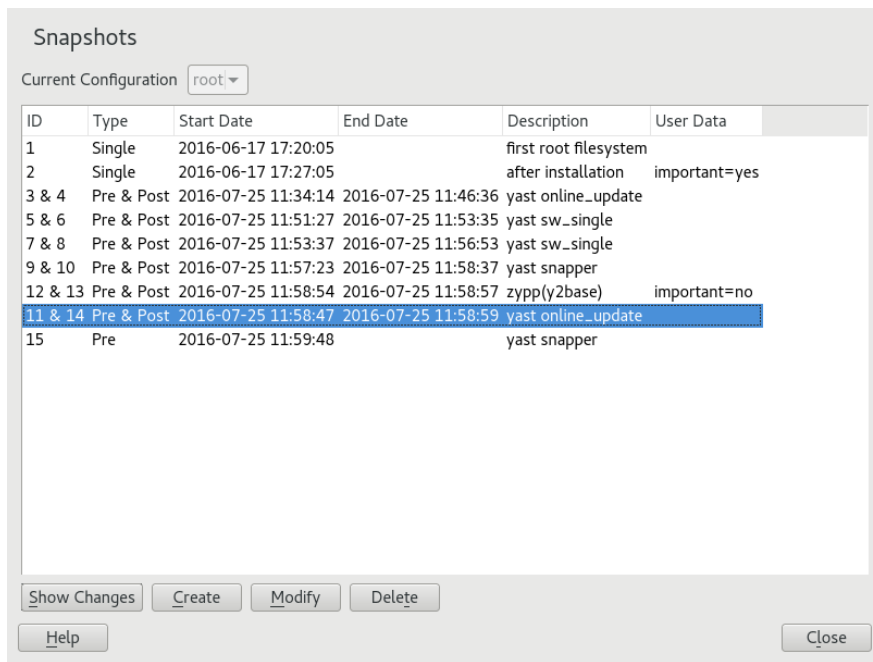
6.2.1 Undoing YaST and Zypper Changes

If you set up the root partition with `Btrfs` during the installation, Snapper—preconfigured for doing rollbacks of YaST or Zypper changes—will automatically be installed. Every time you start a YaST module or a Zypper transaction, two snapshots are created: a “pre-snapshot” capturing the state of the file system before the start of the module and a “post-snapshot” after the module has been finished.

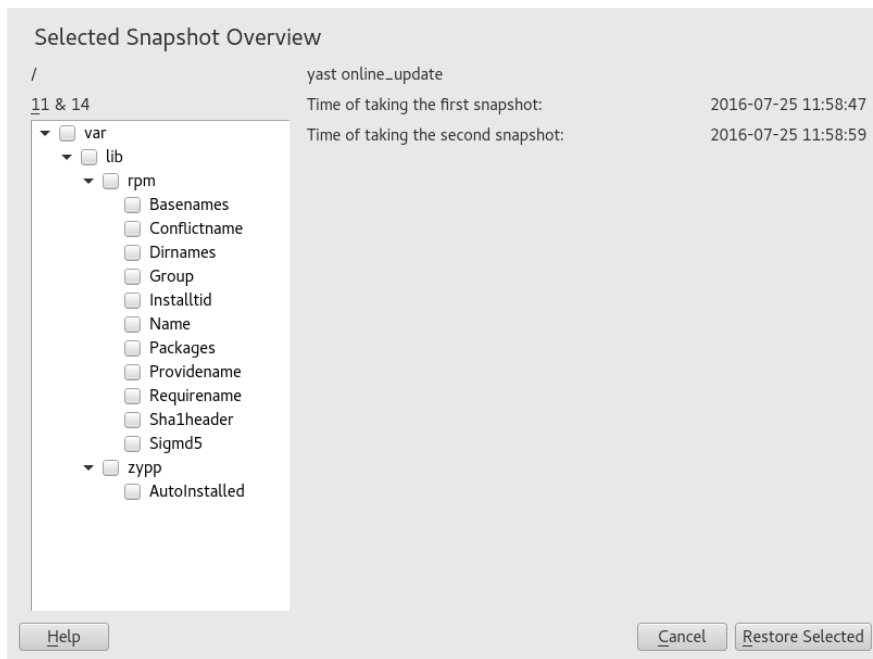
Using the YaST Snapper module or the `snapper` command line tool, you can undo the changes made by YaST/Zypper by restoring files from the “pre-snapshot”. Comparing two snapshots the tools also allow you to see which files have been changed. You can also display the differences between two versions of a file (diff).

PROCEDURE 6.1: UNDOING CHANGES USING THE YAST SNAPPER MODULE

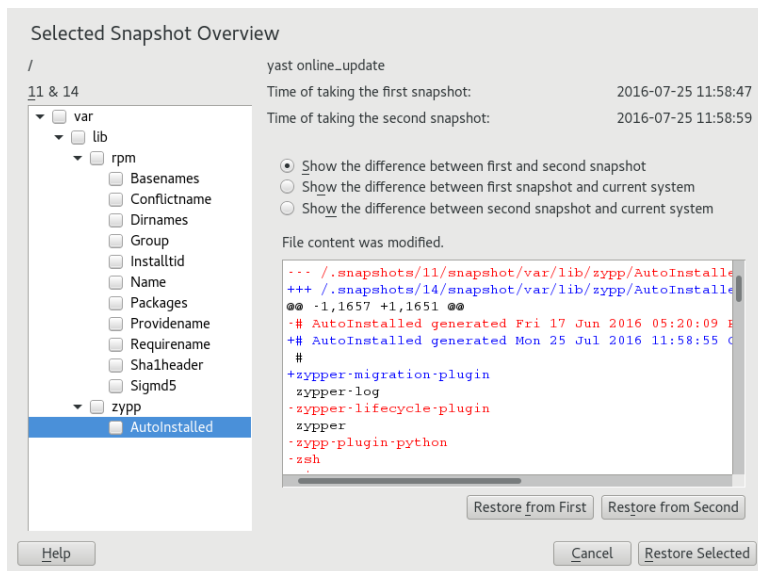
1. Start the *Snapper* module from the *Miscellaneous* section in YaST or by entering `yast2 snapper`.
2. Make sure *Current Configuration* is set to *root*. This is always the case unless you have manually added own Snapper configurations.
3. Choose a pair of pre- and post-snapshots from the list. Both, YaST and Zypper snapshot pairs are of the type *Pre & Post*. YaST snapshots are labeled as `zypp(y2base)` in the *Description column*; Zypper snapshots are labeled `zypp(zypper)`.



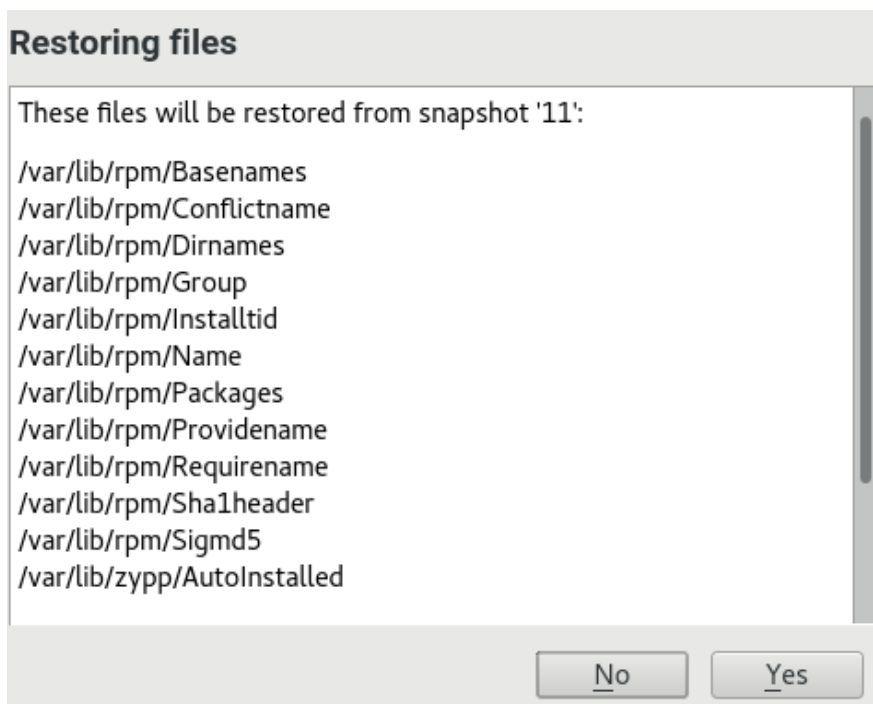
- Click *Show Changes* to open the list of files that differ between the two snapshots.



- Review the list of files. To display a “diff” between the pre- and post-version of a file, select it from the list.



6. To restore one or more files, select the relevant files or directories by activating the respective check box. Click *Restore Selected* and confirm the action by clicking *Yes*.



To restore a single file, activate its diff view by clicking its name. Click *Restore From First* and confirm your choice with *Yes*.

PROCEDURE 6.2: UNDOING CHANGES USING THE **snapper** COMMAND

1. Get a list of YaST and Zypper snapshots by running **`snapper list -t pre-post`**. YaST snapshots are labeled as `yast module_name` in the *Description* column; Zypper snapshots are labeled `zypp(zypper)`.

```
root # snapper list -t pre-post
Pre # | Post # | Pre Date                               | Post Date                               | Description
-----+-----+-----+-----+-----
311   | 312   | Tue 06 May 2014 14:05:46 CEST          | Tue 06 May 2014 14:05:52 CEST          | zypp(y2base)
340   | 341   | Wed 07 May 2014 16:15:10 CEST          | Wed 07 May 2014 16:15:16 CEST          | zypp(zypper)
342   | 343   | Wed 07 May 2014 16:20:38 CEST          | Wed 07 May 2014 16:20:42 CEST          | zypp(y2base)
344   | 345   | Wed 07 May 2014 16:21:23 CEST          | Wed 07 May 2014 16:21:24 CEST          | zypp(zypper)
346   | 347   | Wed 07 May 2014 16:41:06 CEST          | Wed 07 May 2014 16:41:10 CEST          | zypp(y2base)
348   | 349   | Wed 07 May 2014 16:44:50 CEST          | Wed 07 May 2014 16:44:53 CEST          | zypp(y2base)
350   | 351   | Wed 07 May 2014 16:46:27 CEST          | Wed 07 May 2014 16:46:38 CEST          | zypp(y2base)
```

2. Get a list of changed files for a snapshot pair with **`snapper status PRE..POST`**. Files with content changes are marked with `c`, files that have been added are marked with `+` and deleted files are marked with `-`.

```
root # snapper status 350..351
+.... /usr/share/doc/packages/mikachan-fonts
+.... /usr/share/doc/packages/mikachan-fonts/COPYING
+.... /usr/share/doc/packages/mikachan-fonts/dl.html
c.... /usr/share/fonts/truetype/fonts.dir
c.... /usr/share/fonts/truetype/fonts.scale
+.... /usr/share/fonts/truetype/#####-p.ttf
+.... /usr/share/fonts/truetype/#####-pb.ttf
+.... /usr/share/fonts/truetype/#####-ps.ttf
+.... /usr/share/fonts/truetype/#####.ttf
c.... /var/cache/fontconfig/7ef2298fde41cc6eeb7af42e48b7d293-x86_64.cache-4
c.... /var/lib/rpm/Basenames
c.... /var/lib/rpm/Dirnames
c.... /var/lib/rpm/Group
c.... /var/lib/rpm/Installtid
c.... /var/lib/rpm/Name
c.... /var/lib/rpm/Packages
c.... /var/lib/rpm/Providename
c.... /var/lib/rpm/Requirename
c.... /var/lib/rpm/Shalheader
c.... /var/lib/rpm/Sigmd5
```

3. To display the diff for a certain file, run **`snapper diff PRE..POST FILENAME`**. If you do not specify `FILENAME`, a diff for all files will be displayed.

```
root # snapper diff 350..351 /usr/share/fonts/truetype/fonts.scale
```

```

-- /snapshots/350/snapshot/usr/share/fonts/truetype/fonts.scale
2014-04-23 15:58:57.000000000 +0200
+++ /snapshots/351/snapshot/usr/share/fonts/truetype/fonts.scale
2014-05-07 16:46:31.000000000 +0200
@@ -1,4 +1,4 @@
-1174
+1486
ds=y:ai=0.2:luximr.ttf -b&h-luxi mono-bold-i-normal--0-0-0-c-0-iso10646-1
ds=y:ai=0.2:luximr.ttf -b&h-luxi mono-bold-i-normal--0-0-0-c-0-iso8859-1
[...]
```

4. To restore one or more files run **snapper -v undochange** *PRE* *..* *POST* *FILENAMES*. If you do not specify a *FILENAMES*, all changed files will be restored.

```

root # snapper -v undochange 350..351
create:0 modify:13 delete:7
undoing change...
deleting /usr/share/doc/packages/mikachan-fonts
deleting /usr/share/doc/packages/mikachan-fonts/COPYING
deleting /usr/share/doc/packages/mikachan-fonts/dl.html
deleting /usr/share/fonts/truetype/#####-p.ttf
deleting /usr/share/fonts/truetype/#####-pb.ttf
deleting /usr/share/fonts/truetype/#####-ps.ttf
deleting /usr/share/fonts/truetype/#####.ttf
modifying /usr/share/fonts/truetype/fonts.dir
modifying /usr/share/fonts/truetype/fonts.scale
modifying /var/cache/fontconfig/7ef2298fde41cc6eeb7af42e48b7d293-
x86_64.cache-4
modifying /var/lib/rpm/Basenames
modifying /var/lib/rpm/Dirnames
modifying /var/lib/rpm/Group
modifying /var/lib/rpm/Installtid
modifying /var/lib/rpm/Name
modifying /var/lib/rpm/Packages
modifying /var/lib/rpm/Providename
modifying /var/lib/rpm/Requirename
modifying /var/lib/rpm/Shalheader
modifying /var/lib/rpm/Sigmd5
undoing change done
```



Warning: Reverting User Additions

Reverting user additions via undoing changes with Snapper is not recommended. Since certain directories are excluded from snapshots, files belonging to these users will remain in the file system. If a user with the same user ID as a deleted user is created, this user will inherit the files. Therefore it is strongly recommended to use the YaST *User and Group Management* tool to remove users.

6.2.2 Using Snapper to Restore Files

Apart from the installation and administration snapshots, Snapper creates timeline snapshots. You can use these backup snapshots to restore files that have accidentally been deleted or to restore a previous version of a file. By using Snapper's diff feature you can also find out which modifications have been made at a certain point of time.

Being able to restore files is especially interesting for data, which may reside on subvolumes or partitions for which snapshots are not taken by default. To be able to restore files from home directories, for example, create a separate Snapper configuration for /home doing automatic timeline snapshots. See [Section 6.4, "Creating and Modifying Snapper Configurations"](#) for instructions.



Warning: Restoring Files Compared to Rollback

Snapshots taken from the root file system (defined by Snapper's root configuration), can be used to do a system rollback. The recommended way to do such a rollback is to boot from the snapshot and then perform the rollback. See [Section 6.3, "System Rollback by Booting from Snapshots"](#) for details.

Performing a rollback would also be possible by restoring all files from a root file system snapshot as described below. However, this is not recommended. You may restore single files, for example a configuration file from the /etc directory, but not the complete list of files from the snapshot.

This restriction only affects snapshots taken from the root file system!

PROCEDURE 6.3: RESTORING FILES USING THE YAST SNAPPER MODULE

1. Start the *Snapper* module from the *Miscellaneous* section in YaST or by entering **yast2 snapper**.

2. Choose the *Current Configuration* from which to choose a snapshot.
3. Select a timeline snapshot from which to restore a file and choose *Show Changes*. Timeline snapshots are of the type *Single* with a description value of *timeline*.
4. Select a file from the text box by clicking the file name. The difference between the snapshot version and the current system is shown. Activate the check box to select the file for restore. Do so for all files you want to restore.
5. Click *Restore Selected* and confirm the action by clicking *Yes*.

PROCEDURE 6.4: RESTORING FILES USING THE **snapper** COMMAND

1. Get a list of timeline snapshots for a specific configuration by running the following command:

```
snapper -c CONFIG list -t single | grep timeline
```

CONFIG needs to be replaced by an existing Snapper configuration. Use **snapper list-configs** to display a list.

2. Get a list of changed files for a given snapshot by running the following command:

```
snapper -c CONFIG status SNAPSHOT_ID..0
```

Replace SNAPSHOT_ID by the ID for the snapshot from which you want to restore the file(s).

3. Optionally list the differences between the current file version and the one from the snapshot by running

```
snapper -c CONFIG diff SNAPSHOT_ID..0 FILE NAME
```

If you do not specify <FILE NAME>, the difference for all files are shown.

4. To restore one or more files, run

```
snapper -c CONFIG -v undochange  
    SNAPSHOT_ID..0 FILENAME1 FILENAME2
```

If you do not specify file names, all changed files will be restored.

6.3 System Rollback by Booting from Snapshots

The GRUB 2 version included on SUSE Linux Enterprise Server can boot from Btrfs snapshots. Together with Snapper's rollback feature, this allows to recover a misconfigured system. Only snapshots created for the default Snapper configuration (`root`) are bootable.

! Important: Supported Configuration

As of SUSE Linux Enterprise Server 12 SP2 system rollbacks are only supported if the default subvolume configuration of the root partition has not been changed.

When booting a snapshot, the parts of the file system included in the snapshot are mounted read-only; all other file systems and parts that are excluded from snapshots are mounted read-write and can be modified.

! Important: Undoing Changes Compared to Rollback

When working with snapshots to restore data, it is important to know that there are two fundamentally different scenarios Snapper can handle:

Undoing Changes

When undoing changes as described in [Section 6.2, "Using Snapper to Undo Changes"](#), two snapshots are compared and the changes between these two snapshots are reverted. Using this method also allows to explicitly exclude selected files from being restored.

Rollback

When doing rollbacks as described in the following, the system is reset to the state at which the snapshot was taken.

To do a rollback from a bootable snapshot, the following requirements must be met. When doing a default installation, the system is set up accordingly.

REQUIREMENTS FOR A ROLLBACK FROM A BOOTABLE SNAPSHOT

- The root file system needs to be Btrfs. Booting from LVM volume snapshots is not supported.

- The root file system needs to be on a single device, a single partition and a single subvolume. Directories that are excluded from snapshots such as `/srv` (see [Section 6.1.2, “Directories That Are Excluded from Snapshots”](#) for a full list) may reside on separate partitions.
- The system needs to be bootable via the installed boot loader.

To perform a rollback from a bootable snapshot, do as follows:

1. Boot the system. In the boot menu choose *Bootable snapshots* and select the snapshot you want to boot. The list of snapshots is listed by date—the most recent snapshot is listed first.
2. Log in to the system. Carefully check whether everything works as expected. Note that you cannot write to any directory that is part of the snapshot. Data you write to other directories will *not* get lost, regardless of what you do next.
3. Depending on whether you want to perform the rollback or not, choose your next step:
 - a. If the system is in a state where you do not want to do a rollback, reboot to boot into the current system state, to choose a different snapshot, or to start the rescue system.
 - b. If you want to perform the rollback, run

```
sudo snapper rollback
```

and reboot afterward. On the boot screen, choose the default boot entry to reboot into the reinstated system.



Tip: Rolling Back to a Specific Installation State

If snapshots are not disabled during installation, an initial bootable snapshot is created at the end of the initial system installation. You can go back to that state at any time by booting this snapshot. The snapshot can be identified by the description after installation.

A bootable snapshot is also created when starting a system upgrade to a service pack or a new major release (provided snapshots are not disabled).

6.3.1 Accessing and Identifying Snapshot Boot Entries

To boot from a snapshot, reboot your machine and choose *Start Bootloader from a read-only snapshot*. A screen listing all bootable snapshots opens. The most recent snapshot is listed first, the oldest last. Use the keys `↓` and `↑` to navigate and press `Enter` to activate the selected snapshot. Activating a snapshot from the boot menu does not reboot the machine immediately, but rather opens the boot loader of the selected snapshot.



FIGURE 6.1: BOOT LOADER: SNAPSHOTS

Each snapshot entry in the boot loader follows a naming scheme which makes it possible to identify it easily:

```
[*] ① OS ② (KERNEL ③ ,DATE ④ TIME ⑤ ,DESCRIPTION ⑥ )
```

- ① If the snapshot was marked important, the entry is marked with a *.
- ② Operating system label.
- ④ Date in the format YYYY-MM-DD.
- ⑤ Time in the format HH:MM.

- 6 This field contains a description of the snapshot. In case of a manually created snapshot this is the string created with the option `--description` or a custom string (see *Tip: Setting a Custom Description for Boot Loader Snapshot Entries*). In case of an automatically created snapshot, it is the tool that was called, for example `zypp(zypper)` or `yast_sw_single`. Long descriptions may be truncated, depending on the size of the boot screen.



Tip: Setting a Custom Description for Boot Loader Snapshot Entries

It is possible to replace the default string in the description field of a snapshot with a custom string. This is for example useful if an automatically created description is not sufficient, or a user-provided description is too long. To set a custom string `STRING` for snapshot `NUMBER`, use the following command:

```
snapper modify --userdata "bootloader=STRING" NUMBER
```

The description should be no longer than 25 characters—everything that exceeds this size will not be readable on the boot screen.

6.3.2 Limitations

A *complete* system rollback, restoring the complete system to the identical state as it was in when a snapshot was taken, is not possible.

6.3.2.1 Directories Excluded from Snapshots

Root file system snapshots do not contain all directories. See *Section 6.1.2, “Directories That Are Excluded from Snapshots”* for details and reasons. As a general consequence, data from these directories is not restored, resulting in the following limitations.

Add-ons and Third Party Software may be Unusable after a Rollback

Applications and add-ons installing data in subvolumes excluded from the snapshot, such as `/opt`, may not work after a rollback, if others parts of the application data are also installed on subvolumes included in the snapshot. Re-install the application or the add-on to solve this problem.

File Access Problems

If an application had changed file permissions and/or ownership in between snapshot and current system, the application may not be able to access these files. Reset permissions and/or ownership for the affected files after the rollback.

Incompatible Data Formats

If a service or an application has established a new data format in between snapshot and current system, the application may not be able to read the affected data files after a rollback.

Subvolumes with a Mixture of Code and Data

Subvolumes like /srv may contain a mixture of code and data. A rollback may result in non-functional code. A downgrade of the PHP version, for example, may result in broken PHP scripts for the Web server.

User Data

If a rollback removes users from the system, data that is owned by these users in directories excluded from the snapshot, is not removed. If a user with the same user ID is created, this user will inherit the files. Use a tool like find to locate and remove orphaned files.

6.3.2.2 No Rollback of Boot Loader Data

A rollback of the boot loader is not possible, since all “stages” of the boot loader must fit together. This cannot be guaranteed when doing rollbacks of /boot.

6.4 Creating and Modifying Snapper Configurations

The way Snapper behaves is defined in a configuration file that is specific for each partition or Btrfs subvolume. These configuration files reside under /etc/snapper/configs/.

In case the root file system is big enough (approximately 16 GB), snapshots are automatically enabled for the root file system / upon the installation. The corresponding default configuration is named root. It creates and manages the YaST and Zypper snapshot. See [Section 6.4.1.1, “Configuration Data”](#) for a list of the default values.

You may create your own configurations for other partitions formatted with `Btrfs` or existing subvolumes on a `Btrfs` partition. In the following example we will set up a `Snapper` configuration for backing up the Web server data residing on a separate, `Btrfs`-formatted partition mounted at `/srv/www`.

After a configuration has been created, you can either use `snapper` itself or the YaST *Snapper* module to restore files from these snapshots. In YaST you need to select your *Current Configuration*, while you need to specify your configuration for `snapper` with the global switch `-c` (for example, `snapper -c myconfig list`).

To create a new `Snapper` configuration, run `snapper create-config`:

```
snapper -c www-data ❶ create-config /srv/www ❷
```

❶ Name of configuration file.

❷ Mount point of the partition or `Btrfs` subvolume on which to take snapshots.

This command will create a new configuration file `/etc/snapper/configs/www-data` with reasonable default values (taken from `/etc/snapper/config-templates/default`). Refer to [Section 6.4.1, “Managing Existing Configurations”](#) for instructions on how to adjust these defaults.



Tip: Configuration Defaults

Default values for a new configuration are taken from `/etc/snapper/config-templates/default`. To use your own set of defaults, create a copy of this file in the same directory and adjust it to your needs. To use it, specify the `-t` option with the `create-config` command:

```
snapper -c www-data create-config -t my_defaults /srv/www
```

6.4.1 Managing Existing Configurations

The `snapper` offers several subcommands for managing existing configurations. You can list, show, delete and modify them:

List Configurations

Use the command `snapper list-configs` to get all existing configurations:

```
root # snapper list-configs
```

Config	Subvolume
root	/
usr	/usr
local	/local

Show a Configuration

Use the subcommand **snapper -c CONFIG get-config** to display the specified configuration. *Config* needs to be replaced by a configuration name shown by **snapper list-configs**. See [Section 6.4.1.1, "Configuration Data"](#) for more information on the configuration options.

To display the default configuration run

```
snapper -c root get-config
```

Modify a Configuration

Use the subcommand **snapper -c CONFIG set-config OPTION=VALUE** to modify an option in the specified configuration. *Config* needs to be replaced by a configuration name shown by **snapper list-configs**. Possible values for *OPTION* and *VALUE* are listed in [Section 6.4.1.1, "Configuration Data"](#).

Delete a Configuration

Use the subcommand **snapper -c CONFIG delete-config** to delete a configuration. *Config* needs to be replaced by a configuration name shown by **snapper list-configs**.

6.4.1.1 Configuration Data

Each configuration contains a list of options that can be modified from the command line. The following list provides details for each option. To change a value, run **snapper -c CONFIG set-config "KEY=VALUE"**.

ALLOW_GROUPS, ALLOW_USERS

Granting permissions to use snapshots to regular users. See [Section 6.4.1.2, "Using Snapper as Regular User"](#) for more information.

The default value is `" "`.

BACKGROUND_COMPARISON

Defines whether pre and post snapshots should be compared in the background after creation.

The default value is "yes".

EMPTY_*

Defines the clean-up algorithm for snapshots pairs with identical pre and post snapshots. See [Section 6.6.3, "Cleaning Up Snapshot Pairs That Do Not Differ"](#) for details.

FSTYPE

File system type of the partition. Do not change.
The default value is "btrfs".

NUMBER_*

Defines the clean-up algorithm for installation and admin snapshots. See [Section 6.6.1, "Cleaning Up Numbered Snapshots"](#) for details.

QGROUP / SPACE_LIMIT

Adds quota support to the clean-up algorithms. See [Section 6.6.5, "Adding Disk Quota Support"](#) for details.

SUBVOLUME

Mount point of the partition or subvolume to snapshot. Do not change.
The default value is "/".

SYNC_ACL

If Snapper is to be used by regular users (see [Section 6.4.1.2, "Using Snapper as Regular User"](#)) the users must be able to access the .snapshot directories and to read files within them. If SYNC_ACL is set to yes, Snapper automatically makes them accessible using ACLs for users and groups from the ALLOW_USERS or ALLOW_GROUPS entries.
The default value is "no".

TIMELINE_CREATE

If set to yes, hourly snapshots are created. Valid values: yes, no.
The default value is "no".

TIMELINE_CLEANUP / TIMELINE_LIMIT_*

Defines the clean-up algorithm for timeline snapshots. See [Section 6.6.2, "Cleaning Up Timeline Screenshots"](#) for details.

6.4.1.2 Using Snapper as Regular User

By default Snapper can only be used by root. However, there are cases in which certain groups or users need to be able to create snapshots or undo changes by reverting to a snapshot:

- Web site administrators who want to take snapshots of /srv/www
- Users who want to take a snapshot of their home directory

For these purposes Snapper configurations that grant permissions to users or/and groups can be created. The corresponding .snapshots directory needs to be readable and accessible by the specified users. The easiest way to achieve this is to set the SYNC_ACL option to yes.

PROCEDURE 6.5: ENABLING REGULAR USERS TO USE SNAPPER

Note that all steps in this procedure need to be run by root.

1. If not existing, create a Snapper configuration for the partition or subvolume on which the user should be able to use Snapper. Refer to *Section 6.4, "Creating and Modifying Snapper Configurations"* for instructions. Example:

```
snapper --config web_data create /srv/www
```

2. The configuration file is created under /etc/snapper/configs/CONFIG, where CONFIG is the value you specified with -c/--config in the previous step (for example /etc/snapper/configs/web_data). Adjust it according to your needs; see *Section 6.4.1, "Managing Existing Configurations"* for details.
3. Set values for ALLOW_USERS and/or ALLOW_GROUPS to grant permissions to users and/or groups, respectively. Multiple entries need to be separated by Space. To grant permissions to the user www_admin for example, run:

```
snapper -c web_data set-config "ALLOW_USERS=www_admin" SYNC_ACL="yes"
```

4. The given Snapper configuration can now be used by the specified user(s) and/or group(s). You can test it with the list command, for example:

```
www_admin:~ > snapper -c web_data list
```

6.5 Manually Creating and Managing Snapshots

Snapper is not restricted to creating and managing snapshots automatically by configuration; you can also create snapshot pairs (“before and after”) or single snapshots manually using either the command line tool or the YaST module.

All Snapper operations are carried out for an existing configuration (see [Section 6.4, “Creating and Modifying Snapper Configurations”](#) for details). You can only take snapshots of partitions or volumes for which a configuration exists. By default the system configuration (root) is used. If you want to create or manage snapshots for your own configuration you need to explicitly choose it. Use the *Current Configuration* drop-down box in YaST or specify the -c on the command line (**snapper -c MYCONFIG COMMAND**).

6.5.1 Snapshot Metadata

Each snapshot consists of the snapshot itself and some metadata. When creating a snapshot you also need to specify the metadata. Modifying a snapshot means changing its metadata—you cannot modify its content. Use **snapper list** to show existing snapshots and their metadata:

snapper --config home list

Lists snapshots for the configuration home. To list snapshots for the default configuration (root), use **snapper -c root list** or **snapper list**.

snapper list -a

Lists snapshots for all existing configurations.

snapper list -t pre-post

Lists all pre and post snapshot pairs for the default (root) configuration.

snapper list -t single

Lists all snapshots of the type single for the default (root) configuration.

The following metadata is available for each snapshot:

- **Type:** Snapshot type, see [Section 6.5.1.1, “Snapshot Types”](#) for details. This data cannot be changed.
- **Number:** Unique number of the snapshot. This data cannot be changed.
- **Pre Number:** Specifies the number of the corresponding pre snapshot. For snapshots of type post only. This data cannot be changed.

- **Description:** A description of the snapshot.
- **Userdata:** An extended description where you can specify custom data in the form of a comma-separated key = value list: `reason=testing, project=foo`. This field is also used to mark a snapshot as important (`important=yes`) and to list the user that created the snapshot (`user=tux`).
- **Cleanup-Algorithm:** Cleanup-algorithm for the snapshot, see [Section 6.6, “Automatic Snapshot Clean-Up”](#) for details.

6.5.1.1 Snapshot Types

Snapper knows three different types of snapshots: `pre`, `post`, and `single`. Physically they do not differ, but Snapper handles them differently.

pre

Snapshot of a file system *before* a modification. Each `pre` snapshot has got a corresponding `post` snapshot. Used for the automatic YaST/Zypper snapshots, for example.

post

Snapshot of a file system *after* a modification. Each `post` snapshot has got a corresponding `pre` snapshot. Used for the automatic YaST/Zypper snapshots, for example.

single

Stand-alone snapshot. Used for the automatic hourly snapshots, for example. This is the default type when creating snapshots.

6.5.1.2 Cleanup-algorithms

Snapper provides three algorithms to clean up old snapshots. The algorithms are executed in a daily cron-job. It is possible to define the number of different types of snapshots to keep in the Snapper configuration (see [Section 6.4.1, “Managing Existing Configurations”](#) for details).

number

Deletes old snapshots when a certain snapshot count is reached.

timeline

Deletes old snapshots having passed a certain age, but keeps several hourly, daily, monthly, and yearly snapshots.

empty-pre-post

Deletes pre/post snapshot pairs with empty diffs.

6.5.2 Creating Snapshots

Creating a snapshot is done by running **`snapper create`** or by clicking *Create* in the YaST module *Snapper*. The following examples explain how to create snapshots from the command line. It should be easy to adopt them when using the YaST interface.



Tip: Snapshot Description

You should always specify a meaningful description to later be able to identify its purpose. Even more information can be specified via the user data option.

`snapper create --description "Snapshot for week 2 2014"`

Creates a stand-alone snapshot (type single) for the default (root) configuration with a description. Because no cleanup-algorithm is specified, the snapshot will never be deleted automatically.

`snapper --config home create --description "Cleanup in ~tux"`

Creates a stand-alone snapshot (type single) for a custom configuration named home with a description. Because no cleanup-algorithm is specified, the snapshot will never be deleted automatically.

`snapper --config home create --description "Daily data backup" --cleanup-algorithm timeline>`

Creates a stand-alone snapshot (type single) for a custom configuration named home with a description. The file will automatically be deleted when it meets the criteria specified for the timeline cleanup-algorithm in the configuration.

`snapper create --type pre --print-number --description "Before the Apache config cleanup" --userdata "important=yes"`

Creates a snapshot of the type pre and prints the snapshot number. First command needed to create a pair of snapshots used to save a “before” and “after” state. The snapshot is marked as important.

```
snapper create --type post --pre-number 30 --description "After the Apache config cleanup" --userdata "important=yes"
```

Creates a snapshot of the type `post` paired with the `pre` snapshot number `30`. Second command needed to create a pair of snapshots used to save a “before” and “after” state. The snapshot is marked as important.

```
snapper create --command COMMAND --description "Before and after COMMAND"
```

Automatically creates a snapshot pair before and after running `COMMAND`. This option is only available when using snapper on the command line.

6.5.3 Modifying Snapshot Metadata

Snapper allows you to modify the description, the cleanup algorithm, and the user data of a snapshot. All other metadata cannot be changed. The following examples explain how to modify snapshots from the command line. It should be easy to adopt them when using the YaST interface.

To modify a snapshot on the command line, you need to know its number. Use `snapper list` to display all snapshots and their numbers.

The YaST *Snapper* module already lists all snapshots. Choose one from the list and click *Modify*.

```
snapper modify --cleanup-algorithm "timeline" 10
```

Modifies the metadata of snapshot 10 for the default (`root`) configuration. The cleanup algorithm is set to `timeline`.

```
snapper --config home modify --description "daily backup" --cleanup-algorithm "timeline" 120
```

Modifies the metadata of snapshot 120 for a custom configuration named `home`. A new description is set and the cleanup algorithm is unset.

6.5.4 Deleting Snapshots

To delete a snapshot with the YaST *Snapper* module, choose a snapshot from the list and click *Delete*.

To delete a snapshot with the command line tool, you need to know its number. Get it by running `snapper list`. To delete a snapshot, run `snapper delete NUMBER`.

When deleting snapshots with Snapper, the freed space will be claimed by a Btrfs process running in the background. Thus the visibility and the availability of free space is delayed. In case you need space freed by deleting a snapshot to be available immediately, use the option `--sync` with the delete command.



Tip: Deleting Snapshot Pairs

When deleting a pre snapshot, you should always delete its corresponding post snapshot (and vice versa).

`snapper delete 65`

Deletes snapshot 65 for the default (root) configuration.

`snapper -c home delete 89 90`

Deletes snapshots 89 and 90 for a custom configuration named home .

`snapper delete --sync 23`

Deletes snapshot 23 for the default (root) configuration and makes the freed space available immediately.



Tip: Delete Unreferenced Snapshots

Sometimes the Btrfs snapshot is present but the XML file containing the metadata for Snapper is missing. In this case the snapshot is not visible for Snapper and needs to be deleted manually:

```
btrfs subvolume delete /.snapshots/SNAPSHOTNUMBER/snapshot
rm -rf /.snapshots/SNAPSHOTNUMBER
```



Tip: Old Snapshots Occupy More Disk Space

If you delete snapshots to free space on your hard disk, make sure to delete old snapshots first. The older a snapshot is, the more disk space it occupies.

Snapshots are also automatically deleted by a daily cron-job. Refer to [Section 6.5.1.2, “Cleanup-algorithms”](#) for details.

6.6 Automatic Snapshot Clean-Up

Snapshots occupy disk space and over time the amount of disk space occupied by the snapshots may become large. To prevent disks from running out of space, Snapper offers algorithms to automatically delete old snapshots. These algorithms differentiate between timeline snapshots and numbered snapshots (administration plus installation snapshot pairs). You can specify the number of snapshots to keep for each type.

In addition to that, you can optionally specify a disk space quota, defining the maximum amount of disk space the snapshots may occupy. It is also possible to automatically delete pre and post snapshots pairs that do not differ.

A clean-up algorithm is always bound to a single Snapper configuration, so you need to configure algorithms for each configuration. In case you want to prevent certain snapshots from being automatically deleted, refer to [Q:](#).

The default setup (`root`) is configured to do clean-up for numbered snapshots and empty pre and post snapshot pairs. Quota support is enabled—snapshots may not occupy more than 50% of the available disk space of the root partition. Timeline snapshots are disabled by default, therefore the timeline clean-up algorithm is also disabled.

6.6.1 Cleaning Up Numbered Snapshots

Cleaning up for numbered snapshots—administration plus installation snapshot pairs—is controlled by the following parameters of a Snapper configuration.

NUMBER_CLEANUP

Enables or disables clean-up of installation and admin snapshot pairs. If enabled, snapshot pairs are deleted when the total snapshot count exceeds a number specified with `NUMBER_LIMIT` and/or `NUMBER_LIMIT_IMPORTANT` and an age specified with `NUMBER_MIN_AGE`. Valid values: `yes` (enable), `no` (disable).

The default value is `"yes"`.

Example command to change or set:

```
snapper -c CONFIG set-config "NUMBER_CLEANUP=no"
```

NUMBER_LIMIT / NUMBER_LIMIT_IMPORTANT

Defines how many regular and/or important installation and administration snapshot pairs to keep. Only the youngest snapshots will be kept. Ignored if NUMBER_CLEANUP is set to "no".

The default value is "2-10" for NUMBER_LIMIT and "4-10" for NUMBER_LIMIT_IMPORTANT.

Example command to change or set:

```
snapper -c CONFIG set-config "NUMBER_LIMIT=10"
```



Important: Ranged Compared to Constant Values

In case quota support is enabled (see [Section 6.6.5, "Adding Disk Quota Support"](#)) the limit needs to be specified as a minimum-maximum range, for example 2-10. If quota support is disabled, a constant value, for example 10, needs to be provided, otherwise cleaning-up will fail with an error.

NUMBER_MIN_AGE

Defines the minimum age in seconds a snapshot must have before it can automatically be deleted. Snapshots younger than the value specified here will not be deleted, regardless of how many exist.

The default value is "1800".

Example command to change or set:

```
snapper -c CONFIG set-config "NUMBER_MIN_AGE=864000"
```



Note: Limit and Age

NUMBER_LIMIT, NUMBER_LIMIT_IMPORTANT and NUMBER_MIN_AGE are always evaluated. Snapshots are only deleted when *all* conditions are met.

If you always want to keep the number of snapshots defined with NUMBER_LIMIT* regardless of their age, set NUMBER_MIN_AGE to 0.

EXAMPLE 6.1: KEEP THE LAST 10 IMPORTANT AND REGULAR SNAPSHOTS REGARDLESS OF AGE

```
NUMBER_CLEANUP=yes
NUMBER_LIMIT_IMPORTANT=10
NUMBER_LIMIT=10
```

```
NUMBER_MIN_AGE=0
```

On the other hand, if you do not want to keep snapshots beyond a certain age, set NUMBER_LIMIT* to 0 and provide the age with NUMBER_MIN_AGE.

EXAMPLE 6.2: ONLY KEEP SNAPSHOTS YOUNGER THAN TEN DAYS

```
NUMBER_CLEANUP=yes  
NUMBER_LIMIT_IMPORTANT=0  
NUMBER_LIMIT=0  
NUMBER_MIN_AGE=864000
```

6.6.2 Cleaning Up Timeline Screenshots

Cleaning up for timeline snapshots is controlled by the following parameters of a Snapper configuration.

TIMELINE_CLEANUP

Enables or disables clean-up of timeline snapshots. If enabled, snapshots are deleted when the total snapshot count exceeds a number specified with TIMELINE_LIMIT_* *and* an age specified with TIMELINE_MIN_AGE. Valid values: yes, no.

The default value is "yes".

Example command to change or set:

```
snapper -c CONFIG set-config "TIMELINE_CLEANUP=yes"
```

TIMELINE_LIMIT_DAILY, TIMELINE_LIMIT_HOURLY, TIMELINE_LIMIT_MONTHLY,
TIMELINE_LIMIT_WEEKLY, TIMELINE_LIMIT_YEARLY

Number of snapshots to keep for hour, day, month, week, and year.

The default value for each entry is "10", except for TIMELINE_LIMIT_WEEKLY, which is set to "0" by default.

TIMELINE_MIN_AGE

Defines the minimum age in seconds a snapshot must have before it can automatically be deleted.

The default value is "1800".

EXAMPLE 6.3: EXAMPLE TIMELINE CONFIGURATION

```
TIMELINE_CLEANUP="yes"
TIMELINE_CREATE="yes"
TIMELINE_LIMIT_DAILY="7"
TIMELINE_LIMIT_HOURLY="24"
TIMELINE_LIMIT_MONTHLY="12"
TIMELINE_LIMIT_WEEKLY="4"
TIMELINE_LIMIT_YEARLY="2"
TIMELINE_MIN_AGE="1800"
```

This example configuration enables hourly snapshots which are automatically cleaned up. TIMELINE_MIN_AGE and TIMELINE_LIMIT_* are always both evaluated. In this example, the minimum age of a snapshot before it can be deleted is set to 30 minutes (1800 seconds). Since we create hourly snapshots, this ensures that only the latest snapshots are kept. If TIMELINE_LIMIT_DAILY is set to not zero, this means that the first snapshot of the day is kept, too.

SNAPSHOTS TO BE KEPT

- Hourly: The last 24 snapshots that have been made.
- Daily: The first daily snapshot that has been made is kept from the last seven days.
- Monthly: The first snapshot made on the last day of the month is kept for the last twelve months.
- Weekly: The first snapshot made on the last day of the week is kept from the last four weeks.
- Yearly: The first snapshot made on the last day of the year is kept for the last two years.

6.6.3 Cleaning Up Snapshot Pairs That Do Not Differ

As explained in [Section 6.1.1, “Types of Snapshots”](#), whenever you run a YaST module or execute Zypper, a pre snapshot is created on start-up and a post snapshot is created when exiting. In case you have not made any changes there will be no difference between the pre and post snapshots. Such “empty” snapshot pairs can be automatically be deleted by setting the following parameters in a Snapper configuration:

EMPTY_PRE_POST_CLEANUP

If set to yes, pre and post snapshot pairs that do not differ will be deleted.

The default value is "yes".

EMPTY_PRE_POST_MIN_AGE

Defines the minimum age in seconds a pre and post snapshot pair that does not differ must have before it can automatically be deleted.

The default value is "1800".

6.6.4 Cleaning Up Manually Created Snapshots

Snapper does not offer custom clean-up algorithms for manually created snapshots. However, you can assign the number or timeline clean-up algorithm to a manually created snapshot. If you do so, the snapshot will join the “clean-up queue” for the algorithm you specified. You can specify a clean-up algorithm when creating a snapshot, or by modifying an existing snapshot:

snapper create --description "Test" --cleanup-algorithm number

Creates a stand-alone snapshot (type single) for the default (root) configuration and assigns the number clean-up algorithm.

snapper modify --cleanup-algorithm "timeline" 25

Modifies the snapshot with the number 25 and assigns the clean-up algorithm timeline.

6.6.5 Adding Disk Quota Support

In addition to the number and/or timeline clean-up algorithms described above, Snapper supports quotas. You can define what percentage of the available space snapshots are allowed to occupy. This percentage value always applies to the Btrfs subvolume defined in the respective Snapper configuration.

If Snapper was enabled during the installation, quota support is automatically enabled. In case you manually enable Snapper at a later point in time, you can enable quota support by running **snapper setup-quota**. This requires a valid configuration (see [Section 6.4, “Creating and Modifying Snapper Configurations”](#) for more information).

Quota support is controlled by the following parameters of a Snapper configuration.

QGROUP

The Btrfs quota group used by Snapper. If not set, run **snapper setup-quota**. If already set, only change if you are familiar with **man 8 btrfs-qgroup**. This value is set with **snapper setup-quota** and should not be changed.

SPACE_LIMIT

Limit of space snapshots are allowed to use in fractions of 1 (100%). Valid values range from 0 to 1 (0.1 = 10%, 0.2 = 20%, ...).

The following limitations and guidelines apply:

- Quotas are only activated in *addition* to an existing number and/or timeline clean-up algorithm. If no clean-up algorithm is active, quota restrictions are not applied.
- With quota support enabled, Snapper will perform two clean-up runs if required. The first run will apply the rules specified for number and timeline snapshots. Only if the quota is exceeded after this run, the quota-specific rules will be applied in a second run.
- Even if quota support is enabled, Snapper will always keep the number of snapshots specified with the NUMBER_LIMIT* and TIMELINE_LIMIT* values, even if the quota will be exceeded. It is therefore recommended to specify ranged values (*min-max*) for NUMBER_LIMIT* and TIMELINE_LIMIT* to ensure the quota can be applied.

If, for example, NUMBER_LIMIT=5-20 is set, Snapper will perform a first clean up run and reduce the number of regular numbered snapshots to 20. In case these 20 snapshots exceed the quota, Snapper will delete the oldest ones in a second run until the quota is met. A minimum of five snapshots will always be kept, regardless of the amount of space they occupy.

6.7 Frequently Asked Questions

Q: Why does Snapper Never Show Changes in `/var/log`, `/tmp` and Other Directories?

A: For some directories we decided to exclude them from snapshots. See [Section 6.1.2, “Directories That Are Excluded from Snapshots”](#) for a list and reasons. To exclude a path from snapshots we create a subvolume for that path.

Q: How much disk space is used by snapshots? How to free disk space?

A: Displaying the amount of disk space a snapshot allocates is currently not supported by the `Btrfs` tools. However, if you have quota enabled, it is possible to determine how much space would be freed if *all* snapshots would be deleted:

1. Get the quota group ID (`1/0` in the following example):

```
root # snapper -c root get-config | grep QGROUP
QGROUP          | 1/0
```

2. Rescan the subvolume quotas:

```
btrfs quota rescan -w /
```

3. Show the data of the quota group (`1/0` in the following example):

```
root # btrfs qgroup show / | grep "1/0"
1/0          4.80GiB    108.82MiB
```

The third column shows the amount of space that would be freed when deleting all snapshots (`108.82MiB`).

To free space on a `Btrfs` partition containing snapshots you need to delete unneeded snapshots rather than files. Older snapshots occupy more space than recent ones. See [Section 6.1.3.4, “Controlling Snapshot Archiving”](#) for details.

Doing an upgrade from one service pack to another results in snapshots occupying a lot of disk space on the system subvolumes, because a lot of data gets changed (package updates). Manually deleting these snapshots after they are no longer needed is recommended. See [Section 6.5.4, “Deleting Snapshots”](#) for details.

Q: Can I Boot a Snapshot from the Boot Loader?

A: Yes—refer to [Section 6.3, “System Rollback by Booting from Snapshots”](#) for details.

Q: How to make a snapshot permanent?

A: Currently Snapper does not offer means to prevent a snapshot from being deleted manually. However, you can prevent snapshots from being automatically deleted by clean-up algorithms. Manually created snapshots (see [Section 6.5.2, “Creating Snapshots”](#)) have no clean-up algorithm assigned unless you specify one with `--cleanup-algorithm`. Automatically created snapshots always either have the `number` or `timeline` algorithm assigned. To remove such an assignment from one or more snapshots, proceed as follows:

1. List all available snapshots:

```
snapper list -a
```

2. Memorize the number of the snapshot(s) you want to prevent from being deleted.

3. Run the following command and replace the number placeholders with the number(s) you memorized:

```
snapper modify --cleanup-algorithm "" #1 #2 #n
```

4. Check the result by running `snapper list -a` again. The entry in the column `Cleanup` should now be empty for the snapshots you modified.

Q: Where can I get more information on Snapper?

A: See the Snapper home page at <http://snapper.io/> [↗](#).

7 Remote Access with VNC

Virtual Network Computing (VNC) enables you to control a remote computer via a graphical desktop (as opposed to a remote shell access). VNC is platform-independent and lets you access the remote machine from any operating system.

SUSE Linux Enterprise Server supports two different kinds of VNC sessions: One-time sessions that “live” as long as the VNC connection from the client is kept up, and persistent sessions that “live” until they are explicitly terminated.



Note: Session Types

A machine can offer both kinds of sessions simultaneously on different ports, but an open session cannot be converted from one type to the other.

7.1 The **vncviewer** Client

To connect to a VNC service provided by a server, a client is needed. The default in SUSE Linux Enterprise Server is **vncviewer**, provided by the **tigervnc** package.

7.1.1 Connecting Using the vncviewer CLI

To start your VNC viewer and initiate a session with the server, use the command:

```
vncviewer jupiter.example.com:1
```

Instead of the VNC display number you can also specify the port number with two colons:

```
vncviewer jupiter.example.com::5901
```

7.1.2 Connecting Using the vncviewer GUI

By running **vncviewer** without specifying **--listen** or a host to connect to, it will show a window to ask for connection details. Enter the host into the *VNC server* field like in [Section 7.1.1, “Connecting Using the vncviewer CLI”](#) and click *Connect*.

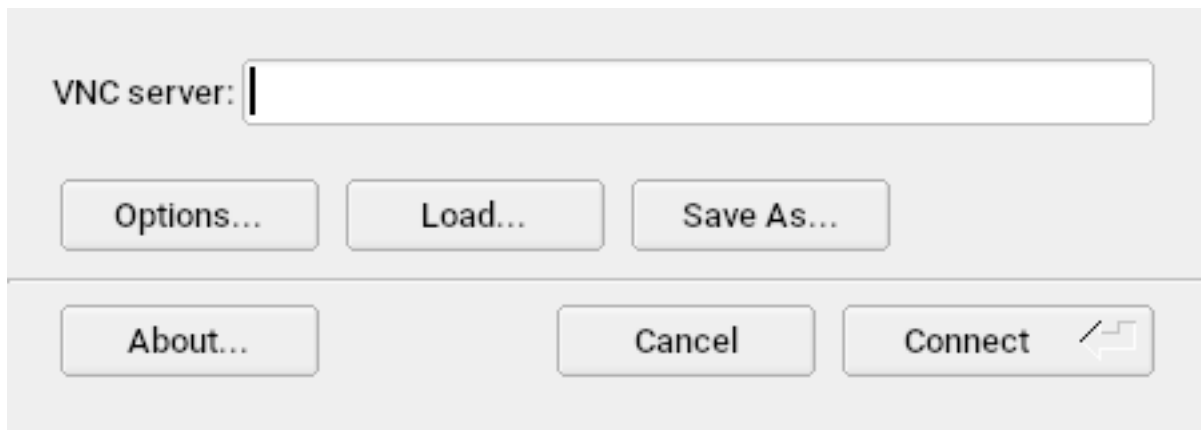


FIGURE 7.1: VNCVIEWER

7.1.3 Notification of Unencrypted Connections

The VNC protocol supports different kinds of encrypted connections, not to be confused with password authentication. If a connection does not use TLS, the text “(Connection not encrypted!)” can be seen in the window title of the VNC viewer.

7.2 One-time VNC Sessions

A one-time session is initiated by the remote client. It starts a graphical login screen on the server. This way you can choose the user which starts the session and, if supported by the login manager, the desktop environment. When you terminate the client connection to such a VNC session, all applications started within that session will be terminated, too. One-time VNC sessions cannot be shared, but it is possible to have multiple sessions on a single host at the same time.

PROCEDURE 7.1: ENABLING ONE-TIME VNC SESSIONS

1. Start *YaST > Network Services > Remote Administration (VNC)*.
2. Check *Allow Remote Administration*.
3. If necessary, also check *Open Port in Firewall* (for example, when your network interface is configured to be in the External Zone). If you have more than one network interface, restrict opening the firewall ports to a specific interface via *Firewall Details*.
4. Confirm your settings with *Finish*.

5. In case not all needed packages are available yet, you need to approve the installation of missing packages.

7.2.1 Available Configurations

The default configuration on SUSE Linux Enterprise Server serves sessions with a resolution of 1024x768 pixels at a color depth of 16-bit. The sessions are available on ports 5901 for “regular” VNC viewers (equivalent to VNC display 1) and on port 5801 for Web browsers.

Other configurations can be made available on different ports, see [Section 7.2.3, “Configuring One-time VNC Sessions”](#).

VNC display numbers and X display numbers are independent in one-time sessions. A VNC display number is manually assigned to every configuration that the server supports (:1 in the example above). Whenever a VNC session is initiated with one of the configurations, it automatically gets a free X display number.

By default, both the VNC client and server try to communicate securely via a self-signed SSL certificate, which is generated after installation. You can either use the default one, or replace it with your own. When using the self-signed certificate, you need to confirm its signature before the first connection—both in the VNC viewer and the Web browser. The Java client is served over HTTPS, using the same certificate as VNC.

7.2.2 Initiating a One-time VNC Session

To connect to a persistent VNC session, a VNC viewer must be installed, see also [Section 7.1, “The vncviewer Client”](#). Alternatively use a Java-capable Web browser to view the VNC session by entering the following URL: <http://jupiter.example.com:5801>

7.2.3 Configuring One-time VNC Sessions

You can skip this section, if you do not need or want to modify the default configuration.

One-time VNC sessions are started via the xinetd daemon. A configuration file is located at /etc/xinetd.d/vnc. By default it offers six configuration blocks: three for VNC viewers (vnc1 to vnc3), and three serving a Java applet (vnchttpd1 to vnchttpd3). By default only vnc1 and vnchttpd1 are active.

To activate a configuration, comment the line `disable = yes` with a `#` character in the first column, or remove that line completely. To deactivate a configuration uncomment or add that line.

The **Xvnc** server can be configured via the `server_args` option—see **Xvnc --help** for a list of options.

When adding custom configurations, make sure they are not using ports that are already in use by other configurations, other services, or existing persistent VNC sessions on the same host.

Activate configuration changes by entering the following command:

```
sudo systemctl reload xinetd
```

Important: Firewall and VNC Ports

When activating Remote Administration as described in *Procedure 7.1, “Enabling One-time VNC Sessions”*, the ports `5801` and `5901` are opened in the firewall. If the network interface serving the VNC sessions is protected by a firewall, you need to manually open the respective ports when activating additional ports for VNC sessions. See *Book “Security Guide”, Chapter 15 “Masquerading and Firewalls”* for instructions.

7.3 Persistent VNC Sessions

A persistent VNC session is initiated on the server. The session and all applications started in this session run regardless of client connections until the session is terminated.

A persistent session can be accessed from multiple clients simultaneously. This is ideal for demonstration purposes where one client has full access and all other clients have view-only access. Another use case are trainings where the trainer might need access to the trainee's desktop. However, most of the times you probably do not want to share your VNC session.

In contrast to one-time sessions that start a display manager, a persistent session starts a ready-to-operate desktop that runs as the user that started the VNC session. Access to persistent sessions is protected by a password.

Access to persistent sessions is protected by two possible types of passwords:

- a regular password that grants full access or
- an optional view-only password that grants a non-interactive (view-only) access.

A session can have multiple client connections of both kinds at once.

PROCEDURE 7.2: STARTING A PERSISTENT VNC SESSION

1. Open a shell and make sure you are logged in as the user that should own the VNC session.
2. If the network interface serving the VNC sessions is protected by a firewall, you need to manually open the port used by your session in the firewall. If starting multiple sessions you may alternatively open a range of ports. See *Book “Security Guide”, Chapter 15 “Masquerading and Firewalls”* for details on how to configure the firewall.
vncserver uses the ports 5901 for display :1, 5902 for display :2, and so on. For persistent sessions, the VNC display and the X display usually have the same number.
3. To start a session with a resolution of 1024x769 pixel and with a color depth of 16-bit, enter the following command:

```
vncserver -geometry 1024x768 -depth 16
```

The vncserver command picks an unused display number when none is given and prints its choice. See man 1 vncserver for more options.

When running vncserver for the first time, it asks for a password for full access to the session. If needed, you can also provide a password for view-only access to the session.

The password(s) you are providing here are also used for future sessions started by the same user. They can be changed with the vncpasswd command.

Important: Security Considerations

Make sure to use strong passwords of significant length (eight or more characters). Do not share these passwords.

VNC connections are unencrypted, so people who can sniff the network(s) between the two machines can read the password when it gets transferred at the beginning of a session.

To terminate the session shut down the desktop environment that runs inside the VNC session from the VNC viewer as you would shut it down if it was a regular local X session.

If you prefer to manually terminate a session, open a shell on the VNC server and make sure you are logged in as the user that owns the VNC session you want to terminate. Run the following command to terminate the session that runs on display `:1`: `vncserver -kill :1`

7.3.1 Connecting to a Persistent VNC Session

To connect to a persistent VNC session, a VNC viewer must be installed, see also [Section 7.1, “The vncviewer Client”](#). Alternatively use a Java-capable Web browser to view the VNC session by entering the following URL: `http://jupiter.example.com:5801`

7.3.2 Configuring Persistent VNC Sessions

Persistent VNC sessions can be configured by editing `$HOME/.vnc/xstartup`. By default this shell script starts the same GUI/window manager it was started from. In SUSE Linux Enterprise Server this will either be GNOME or IceWM. If you want to start your session with a window manager of your choice, set the variable `WINDOWMANAGER`:

```
WINDOWMANAGER=gnome vncserver -geometry 1024x768
WINDOWMANAGER=icewm vncserver -geometry 1024x768
```



Note: One Configuration for Each User

Persistent VNC sessions are configured in a single per-user configuration. Multiple sessions started by the same user will all use the same start-up and password files.

8 File Synchronization

These days, many people use several computers—one computer at home, one or several computers at the workplace, and possibly a laptop, tablet, or a smartphone on the road. Many files are needed on all these computers. You should be able to work with all computers and modify the files so that you have the latest version of the data available on all computers.

8.1 Available Data Synchronization Software

Data synchronization is no problem for computers that are permanently linked by means of a fast network. In this case, use a network file system, like NFS, and store the files on a server, enabling all hosts to access the same data via the network. This approach is impossible if the network connection is poor or not permanent. When you are on the road with a laptop, copies of all needed files must be on the local hard disk. However, it is then necessary to synchronize modified files. When you modify a file on one computer, make sure a copy of the file is updated on all other computers. For occasional copies, this can be done manually with `scp` or `rsync`. However, if many files are involved, the procedure can be complicated and requires great care to avoid errors, such as overwriting a new file with an old file.



Warning: Risk of Data Loss

Before you start managing your data with a synchronization system, you should be well acquainted with the program used and test its functionality. A backup is indispensable for important files.

The time-consuming and error-prone task of manually synchronizing data can be avoided by using one of the programs that use various methods to automate this job. The following summaries are merely intended to convey a general understanding of how these programs work and how they can be used. If you plan to use them, read the program documentation.

These days file synchronization can also be done with a cloud computing solution.

8.1.1 CVS

CVS, which is mostly used for managing program source versions, offers the possibility of keeping copies of the files on multiple computers. Accordingly, it is also suitable for data synchronization. CVS maintains a central repository on the server in which the files and changes to files are saved. Changes that are performed locally are committed to the repository and can be retrieved from other computers by means of an update. Both procedures must be initiated by the user.

CVS is very resilient to errors when changes occur on several computers. The changes are merged and (if changes took place in the same lines) a conflict is reported. When a conflict occurs, the database remains in a consistent state. The conflict is only visible for resolution on the client host.

8.1.2 rsync

When no version control is needed but large directory structures need to be synchronized over slow network connections, the tool rsync offers well-developed mechanisms for transmitting only changes within files. This not only applies to text files, but also binary files. To detect the differences between files, rsync subdivides the files into blocks and computes check sums over them.

The effort put into the detection of the changes comes at a price. The systems to synchronize should be scaled generously for the usage of rsync. RAM is especially important.

8.2 Determining Factors for Selecting a Program

There are some important factors to consider when deciding which program to use.

8.2.1 Client/Server versus Peer-to-Peer

Two different models are commonly used for distributing data. In the first model, all clients synchronize their files with a central server. The server must be accessible by all clients at least occasionally. This model is used by CVS.

The other possibility is to let all networked hosts synchronize their data between each other as peers. rsync actually works in client mode, but any client can also act as a server.

8.2.2 Portability

CVS and rsync are also available for many other operating systems, including various Unix and Windows systems.

8.2.3 Interactive versus Automatic

In CVS, the data synchronization is started manually by the user. This allows fine control over the data to synchronize and easy conflict handling. However, if the synchronization intervals are too long, conflicts are more likely to occur.

8.2.4 Conflicts: Incidence and Solution

Conflicts only rarely occur in CVS, even when several people work on one large program project. This is because the documents are merged on the basis of individual lines. When a conflict occurs, only one client is affected. Usually conflicts in CVS can easily be resolved.

There is no conflict handling in rsync. The user is responsible for not accidentally overwriting files and manually resolving all possible conflicts. To be on the safe side, a versioning system like RCS can additionally be employed.

8.2.5 Selecting and Adding Files

In CVS, new directories and files must be added explicitly using the command `cvs add`. This results in greater user control over the files to synchronize. On the other hand, new files are often overlooked, especially when the question marks in the output of `cvs update` are ignored because of the large number of files.

8.2.6 History

An additional feature of CVS is that old file versions can be reconstructed. A brief editing remark can be inserted for each change and the development of the files can easily be traced later based on the content and the remarks. This is a valuable aid for theses and program texts.

8.2.7 Data Volume and Hard Disk Requirements

A sufficient amount of free space for all distributed data is required on the hard disks of all involved hosts. CVS require additional space for the repository database on the server. The file history is also stored on the server, requiring even more space. When files in text format are changed, only the modified lines need to be saved. Binary files require additional space amounting to the size of the file every time the file is changed.

8.2.8 GUI

Experienced users normally run CVS from the command line. However, graphical user interfaces are available for Linux (such as cervisia) and other operating systems (like wincvs). Many development tools and text editors (such as Emacs) provide support for CVS. The resolution of conflicts is often much easier to perform with these front-ends.

8.2.9 User Friendliness

rsync is rather easy to use and is also suitable for newcomers. CVS is somewhat more difficult to operate. Users should understand the interaction between the repository and local data. Changes to the data should first be merged locally with the repository. This is done with the command `cvs update`. Then the data must be sent back to the repository with the command `cvs com-
mit`. When this procedure has been understood, newcomers are also able to use CVS.

8.2.10 Security against Attacks

During transmission, the data should ideally be protected against interception and manipulation. CVS and rsync can easily be used via SSH (secure shell), providing security against attacks of this kind. Running CVS via rsh (remote shell) should be avoided. Accessing CVS with the *pserver* mechanism in insecure networks is likewise not advisable.

8.2.11 Protection against Data Loss

CVS has been used by developers for a long time to manage program projects and is extremely stable. Because the development history is saved, CVS even provides protection against certain user errors, such as unintentional deletion of a file.

TABLE 8.1: FEATURES OF THE FILE SYNCHRONIZATION TOOLS: -- = VERY POOR, - = POOR OR NOT AVAILABLE, O = MEDIUM, + = GOOD, ++ = EXCELLENT, X = AVAILABLE

	CVS	rsync
Client/Server	C-S	C-S
Portability	Lin,Un*x,Win	Lin,Un*x,Win
Interactivity	x	x
Speed	o	+
Conflicts	+ +	o
File Sel.	Sel./file, dir.	Dir.
History	x	-
Hard Disk Space	--	o
GUI	o	-
Difficulty	o	+
Attacks	+ (SSH)	+ (SSH)
Data Loss	+ +	+

8.3 Introduction to CVS

CVS is suitable for synchronization purposes if individual files are edited frequently and are stored in a file format, such as ASCII text or program source text. The use of CVS for synchronizing data in other formats (such as JPEG files) is possible, but leads to large amounts of data, because all variants of a file are stored permanently on the CVS server. In such cases, most of the capabilities of CVS cannot be used. The use of CVS for synchronizing files is only possible if all workstations can access the same server.

8.3.1 Configuring a CVS Server

The *server* is the host on which all valid files are located, including the latest versions of all files. Any stationary workstation can be used as a server. If possible, the data of the CVS repository should be included in regular backups.

When configuring a CVS server, it might be a good idea to grant users access to the server via SSH. If the user is known to the server as tux and the CVS software is installed on the server and the client, the following environment variables must be set on the client side:

```
CVS_RSH=ssh CVSROOT=tux@server:/serverdir
```

The command **cv**s **init** can be used to initialize the CVS server from the client side. This needs to be done only once.

Finally, the synchronization must be assigned a name. Select or create a directory on the client to contain files to manage with CVS (the directory can also be empty). The name of the directory is also the name of the synchronization. In this example, the directory is called synchome. Change to this directory and enter the following command to set the synchronization name to synchome:

```
cvs import synchome tux wilber
```

Many CVS commands require a comment. For this purpose, CVS starts an editor (the editor defined in the environment variable \$EDITOR or vi if no editor was defined). The editor call can be circumvented by entering the comment in advance on the command line, such as in the following example:

```
cvs import -m 'this is a test' synchome tux wilber
```

8.3.2 Using CVS

The synchronization repository can now be checked out from all hosts with **cv**s **co** synchome. This creates a new subdirectory synchome on the client. To commit your changes to the server, change to the directory synchome (or one of its subdirectories) and enter **cv**s **commit**.

By default, all files (including subdirectories) are committed to the server. To commit only individual files or directories, specify them as in `cvsc commit file1 directory1`. New files and directories must be added to the repository with a command like `cvsc add file1 directory1` before they are committed to the server. Subsequently, commit the newly added files and directories with `cvsc commit file1 directory1`.

If you change to another workstation, check out the synchronization repository if this has not been done during an earlier session at the same workstation.

Start the synchronization with the server with `cvsc update`. Update individual files or directories as in `cvsc update file1 directory1`. To see the difference between the current files and the versions stored on the server, use the command `cvsc diff` or `cvsc diff file1 directory1`. Use `cvsc -nq update` to see which files would be affected by an update.

Here are some status symbols displayed during an update:

U

The local version was updated. This affects all files that are provided by the server and missing on the local system.

M

The local version was modified. If there were changes on the server, it was possible to merge the differences in the local copy.

P

The local version was patched with the version on the server.

C

The local file conflicts with current version in the repository.

?

This file does not exist in CVS.

The status **M** indicates a locally modified file. Either commit the local copy to the server or remove the local file and run the update again. In this case, the missing file is retrieved from the server. If you commit a locally modified file and the file was changed in the same line and committed, you might get a conflict, indicated with **C**.

In this case, look at the conflict marks (“>>” and “<<”) in the file and decide between the two versions. As this can be a rather unpleasant job, you might decide to abandon your changes, delete the local file, and enter `cvs up` to retrieve the current version from the server.

8.4 Introduction to rsync

rsync is useful when large amounts of data need to be transmitted regularly while not changing too much. This is, for example, often the case when creating backups. Another application concerns staging servers. These are servers that store complete directory trees of Web servers that are regularly mirrored onto a Web server in a DMZ.

8.4.1 Configuration and Operation

rsync can be operated in two different modes. It can be used to archive or copy data. To accomplish this, only a remote shell, like SSH, is required on the target system. However, rsync can also be used as a daemon to provide directories to the network.

The basic mode of operation of rsync does not require any special configuration. rsync directly allows mirroring complete directories onto another system. As an example, the following command creates a backup of the home directory of `tux` on a backup server named `sun`:

```
rsync -baz -e ssh /home/tux/ tux@sun:backup
```

The following command is used to play the directory back:

```
rsync -az -e ssh tux@sun:backup /home/tux/
```

Up to this point, the handling does not differ much from that of a regular copying tool, like `scp`. rsync should be operated in “rsync” mode to make all its features fully available. This is done by starting the `rsyncd` daemon on one of the systems. Configure it in the file `/etc/rsyncd.conf`. For example, to make the directory `/srv/ftp` available with rsync, use the following configuration:

```
gid = nobody
uid = nobody
read only = true
```

```
use chroot = no
transfer logging = true
log format = %h %o %f %l %b
log file = /var/log/rsyncd.log

[FTP]
    path = /srv/ftp
    comment = An Example
```

Then start rsyncd with **`systemctl start rsyncd`**. rsyncd can also be started automatically during the boot process. Set this up by activating this service in the YaST *Services Manager* or by manually entering the command:

```
root # systemctl enable rsyncd
```

syncd can alternatively be started by xinetd. This is, however, only recommended for servers that rarely use rsyncd.

The example also creates a log file listing all connections. This file is stored in `/var/log/rsyncd.log`.

It is then possible to test the transfer from a client system. Do this with the following command:

```
rsync -avz sun::FTP
```


This command lists all files present in the directory `/srv/ftp` of the server. This request is also logged in the log file `/var/log/rsyncd.log`. To start an actual transfer, provide a target directory. Use `.` for the current directory. For example:

```
rsync -avz sun::FTP .
```

By default, no files are deleted while synchronizing with rsync. If this should be forced, the additional option `--delete` must be stated. To ensure that no newer files are deleted, the option `--update` can be used instead. Any conflicts that arise must be resolved manually.

8.5 For More Information

CVS

Important information about CVS can be found in the home page <http://www.cvshome.org> .

rsync

Important information about rsync is provided in the man pages [man rsync](#) and [man rsyncd.conf](#). A technical reference about the operating principles of rsync is featured in [/usr/share/doc/packages/rsync/tech_report.ps](#). Find the latest news about rsync on the project Web site at <http://rsync.samba.org/>.

Subversion

Subversion is available from the SUSE Linux Enterprise SDK. The SDK is a module for SUSE Linux Enterprise and is available via an online channel from the SUSE Customer Center. Alternatively, go to <http://download.suse.com/>, search for [SUSE Linux Enterprise Software Development Kit](#) and download it from there. Refer to *Book “Deployment Guide”, Chapter 13 “Installing Modules, Extensions, and Third Party Add-On Products”* for details.

II System

- 9 32-Bit and 64-Bit Applications in a 64-Bit System Environment **116**
- 10 Booting a Linux System **121**
- 11 The Boot Loader GRUB 2 **127**
- 12 UEFI (Unified Extensible Firmware Interface) **146**
- 13 The systemd Daemon **156**
- 14 **journalctl**: Query the systemd Journal **179**
- 15 Basic Networking **187**
- 16 Printer Operation **256**
- 17 The X Window System **271**
- 18 Accessing File Systems with FUSE **285**
- 19 Dynamic Kernel Device Management with udev **287**
- 20 Live Patching the Linux Kernel Using kGraft **299**
- 21 Special System Features **305**

9 32-Bit and 64-Bit Applications in a 64-Bit System Environment

SUSE® Linux Enterprise Server is available for several 64-bit platforms. This does not necessarily mean that all the applications included have already been ported to 64-bit platforms. SUSE Linux Enterprise Server supports the use of 32-bit applications in a 64-bit system environment. This chapter offers a brief overview of how this support is implemented on 64-bit SUSE Linux Enterprise Server platforms. It explains how 32-bit applications are executed (runtime support) and how 32-bit applications should be compiled to enable them to run both in 32-bit and 64-bit system environments. Additionally, find information about the kernel API and an explanation of how 32-bit applications can run under a 64-bit kernel.

SUSE Linux Enterprise Server for the 64-bit platforms POWER, z Systems and AMD64/Intel 64 is designed so that existing 32-bit applications run in the 64-bit environment “out-of-the-box.” The corresponding 32-bit platforms are ppc for POWER, and x86 for AMD64/Intel 64. This support means that you can continue to use your preferred 32-bit applications without waiting for a corresponding 64-bit port to become available. The current POWER system runs most applications in 32-bit mode, but you can run 64-bit applications.

9.1 Runtime Support



Important: Conflicts Between Application Versions

If an application is available both for 32-bit and 64-bit environments, parallel installation of both versions is bound to lead to problems. In such cases, decide on one of the two versions and install and use this.

An exception to this rule is PAM (pluggable authentication modules). SUSE Linux Enterprise Server uses PAM in the authentication process as a layer that mediates between user and application. On a 64-bit operating system that also runs 32-bit applications it is necessary to always install both versions of a PAM module.

To be executed correctly, every application requires a range of libraries. Unfortunately, the names for the 32-bit and 64-bit versions of these libraries are identical. They must be differentiated from each other in another way.

To retain compatibility with the 32-bit version, the libraries are stored at the same place in the system as in the 32-bit environment. The 32-bit version of `libc.so.6` is located under `/lib/libc.so.6` in both the 32-bit and 64-bit environments.

All 64-bit libraries and object files are located in directories called `lib64`. The 64-bit object files that you would normally expect to find under `/lib` and `/usr/lib` are now found under `/lib64` and `/usr/lib64`. This means that there is space for the 32-bit libraries under `/lib` and `/usr/lib`, so the file name for both versions can remain unchanged.

Subdirectories of 32-bit `/lib` directories which contain data content that does not depend on the word size are not moved. This scheme conforms to LSB (Linux Standards Base) and FHS (File System Hierarchy Standard).

9.2 Software Development

All 64-bit architectures support the development of 64-bit objects. The level of support for 32-bit compiling depends on the architecture. These are the various implementation options for the toolchain from GCC (GNU Compiler Collection) and binutils, which include the assembler `as` and the linker `ld`:

Both 32-bit and 64-bit objects can be generated with a biarch development toolchain. A biarch development toolchain allows generation of 32-bit and 64-bit objects. The compilation of 64-bit objects is the default on almost all platforms. 32-bit objects can be generated if special flags are used. This special flag is `-m32` for GCC. The flags for the binutils are architecture-dependent, but GCC transfers the correct flags to linkers and assemblers. A biarch development toolchain currently exists for amd64 (supports development for x86 and amd64 instructions), for z Systems and for POWER. 32-bit objects are normally created on the POWER platform. The `-m64` flag must be used to generate 64-bit objects.

All header files must be written in an architecture-independent form. The installed 32-bit and 64-bit libraries must have an API (application programming interface) that matches the installed header files. The normal SUSE Linux Enterprise Server environment is designed according to this principle. In the case of manually updated libraries, resolve these issues yourself.

9.3 Software Compilation on Biarch Platforms

To develop binaries for the other architecture on a biarch architecture, the respective libraries for the second architecture must additionally be installed. These packages are called `rpmname-32bit` or `rpmname-x86` if the second architecture is a 32-bit architecture or `rpmname-64bit` if the second architecture is a 64-bit architecture. You also need the respective headers and libraries from the `rpmname-devel` packages and the development libraries for the second architecture from `rpmname-devel-32bit` or `rpmname-devel-64bit`.

For example, to compile a program that uses **libaio** on a system whose second architecture is a 32-bit architecture (x86_64 or z Systems), you need the following RPMs:

libaio-32bit

32-bit runtime package

libaio-devel-32bit

Headers and libraries for 32-bit development

libaio

64-bit runtime package

libaio-devel

64-bit development headers and libraries

Most open source programs use an **autoconf**-based program configuration. To use **autoconf** for configuring a program for the second architecture, overwrite the normal compiler and linker settings of **autoconf** by running the **configure** script with additional environment variables. The following example refers to an x86_64 system with x86 as the second architecture. Examples for POWER with ppc as the second architecture would be similar.

1. Use the 32-bit compiler:

```
CC="gcc -m32"
```

2. Instruct the linker to process 32-bit objects (always use **gcc** as the linker front-end):

```
LD="gcc -m32"
```

3. Set the assembler to generate 32-bit objects:

```
AS="gcc -c -m32"
```

4. Specify linker flags, such as the location of 32-bit libraries, for example:

```
LDFLAGS="-L/usr/lib"
```

5. Specify the location for the 32-bit object code libraries:

```
--libdir=/usr/lib
```

6. Specify the location for the 32-bit X libraries:

```
--x-libraries=/usr/lib
```

Not all of these variables are needed for every program. Adapt them to the respective program. An example **`configure`** call to compile a native 32-bit application on x86_64, POWER or z Systems could appear as follows:

```
CC="gcc -m32"  
LDFLAGS="-L/usr/lib;"  
./configure --prefix=/usr --libdir=/usr/lib --x-libraries=/usr/lib  
make  
make install
```

9.4 Kernel Specifications

The 64-bit kernels for AMD64/Intel 64, POWER and z Systems offer both a 64-bit and a 32-bit kernel ABI (application binary interface). The latter is identical with the ABI for the corresponding 32-bit kernel. This means that the 32-bit application can communicate with the 64-bit kernel in the same way as with the 32-bit kernel.

The 32-bit emulation of system calls for a 64-bit kernel does not support all the APIs used by system programs. This depends on the platform. For this reason, few applications, like **`lspci`**, must be compiled on non-POWER platforms as 64-bit programs to function properly. On IBM z Systems, not all ioctls are available in the 32-bit kernel ABI.

A 64-bit kernel can only load 64-bit kernel modules that have been specially compiled for this kernel. It is not possible to use 32-bit kernel modules.



Tip: Kernel-loadable Modules

Some applications require separate kernel-loadable modules. If you intend to use such a 32-bit application in a 64-bit system environment, contact the provider of this application and SUSE to make sure that the 64-bit version of the kernel-loadable module and the 32-bit compiled version of the kernel API are available for this module.

10 Booting a Linux System

Booting a Linux system involves different components and tasks. The hardware itself is initialized by the BIOS or the UEFI, which starts the Kernel by means of a boot loader. After this point, the boot process is completely controlled by the operating system and handled by `systemd`. `systemd` provides a set of “targets” that boot setups for everyday usage, maintenance or emergencies.

10.1 The Linux Boot Process

The Linux boot process consists of several stages, each represented by a different component. The following list briefly summarizes the boot process and features all the major components involved:

1. **BIOS/UEFI.** After turning on the computer, the BIOS or the UEFI initializes the screen and keyboard, and tests the main memory. Up to this stage, the machine does not access any mass storage media. Subsequently, the information about the current date, time, and the most important peripherals are loaded from the CMOS values. When the first hard disk and its geometry are recognized, the system control passes from the BIOS to the boot loader. If the BIOS supports network booting, it is also possible to configure a boot server that provides the boot loader. On AMD64/Intel 64 systems, PXE boot is needed. Other architectures commonly use the BOOTP protocol to get the boot loader.
2. **Boot Loader.** The first physical 512-byte data sector of the first hard disk is loaded into the main memory and the *boot loader* that resides at the beginning of this sector takes over. The commands executed by the boot loader determine the remaining part of the boot process. Therefore, the first 512 bytes on the first hard disk are called the *Master Boot Record* (MBR). The boot loader then passes control to the actual operating system, in this case, the Linux Kernel. More information about GRUB 2, the Linux boot loader, can be found in [Chapter 11, The Boot Loader GRUB 2](#). For a network boot, the BIOS acts as the boot loader. It gets the boot image from the boot server and starts the system. This is completely independent of local hard disks.

If the root file system fails to mount from within the boot environment, it must be checked and repaired before the boot can continue. The file system checker will be automatically started for Ext3 and Ext4 file systems. The repair process is not automated for XFS and

Btrfs file systems and the user will be presented with information describing the options available to repair the file system. Once the file system has been successfully repaired, exiting the boot environment will cause the system to retry mounting the root file system and, if successful, the boot will continue normally.

3. **Kernel and initramfs.** To pass system control, the boot loader loads both the Kernel and an initial RAM-based file system (initramfs) into memory. The contents of the initramfs can be used by the Kernel directly. initramfs contains a small executable called init that handles the mounting of the real root file system. If special hardware drivers are needed before the mass storage can be accessed, they must be in initramfs. For more information about initramfs, refer to [Section 10.2, “initramfs”](#). If the system does not have a local hard disk, the initramfs must provide the root file system for the Kernel. This can be done using a network block device like iSCSI or SAN, but it is also possible to use NFS as the root device.



Note: The init Process Naming

Two different programs are commonly named “init”:

- a. the initramfs process mounting the root file system
- b. the operating system process setting up the system

In this chapter we will therefore refer to them as “init on initramfs” and “systemd”, respectively.

4. **init on initramfs.** This program performs all actions needed to mount the proper root file system. It provides Kernel functionality for the needed file system and device drivers for mass storage controllers with udev. After the root file system has been found, it is checked for errors and mounted. If this is successful, the initramfs is cleaned and the systemd daemon on the root file system is executed. For more information about init on initramfs, refer to [Section 10.3, “Init on initramfs”](#). Find more information about udev in [Chapter 19, Dynamic Kernel Device Management with udev](#).
5. **systemd.** By starting services and mounting file systems, systemd handles the actual booting of the system. systemd is described in [Chapter 13, The systemd Daemon](#).

10.2 `initramfs`

`initramfs` is a small `cpio` archive that the Kernel can load into a RAM disk. It provides a minimal Linux environment that enables the execution of programs before the actual root file system is mounted. This minimal Linux environment is loaded into memory by BIOS or UEFI routines and does not have specific hardware requirements other than sufficient memory. The `initramfs` archive must always provide an executable named `init` that executes the `systemd` daemon on the root file system for the boot process to proceed.

Before the root file system can be mounted and the operating system can be started, the Kernel needs the corresponding drivers to access the device on which the root file system is located. These drivers may include special drivers for certain kinds of hard disks or even network drivers to access a network file system. The needed modules for the root file system may be loaded by `init` on `initramfs`. After the modules are loaded, `udev` provides the `initramfs` with the needed devices. Later in the boot process, after changing the root file system, it is necessary to regenerate the devices. This is done by the `systemd` unit `udev.service` with the command `udevtrigger`.

If you need to change hardware (for example, hard disks) in an installed system and this hardware requires different drivers to be in the Kernel at boot time, you must update the `initramfs` file. This is done by calling `dracut -f` (the option `-f` overwrites the existing `initramfs` file). To add a driver for the new hardware, edit `/etc/dracut.conf.d/01-dist.conf` and add the following line.

```
force_drivers+="driver1"
```

Replace `driver1` with the module name of the driver. If you need to add more than one driver, list them space-separated (`driver1 driver2`).

Important: Updating `initramfs` or `init`

The boot loader loads `initramfs` or `init` in the same way as the Kernel. It is not necessary to re-install GRUB 2 after updating `initramfs` or `init`, because GRUB 2 searches the directory for the right file when booting.



Tip: Changing Kernel Variables

If you change the values of some kernel variables via the `sysctl` interface by editing related files (`/etc/sysctl.conf` or `/etc/sysctl.d/*.conf`), the change will be lost on the next system reboot. Even if you load the values with `sysctl --system` at runtime, the changes are not saved into the `initramfs` file. You need to update it by calling `dracut -f` (the option `-f` overwrites the existing `initramfs` file).

10.3 Init on `initramfs`

The main purpose of `init` on `initramfs` is to prepare the mounting of and access to the real root file system. Depending on your system configuration, `init` on `initramfs` is responsible for the following tasks.

Loading Kernel Modules

Depending on your hardware configuration, special drivers may be needed to access the hardware components of your computer (the most important component being your hard disk). To access the final root file system, the Kernel needs to load the proper file system drivers.

Providing Block Special Files

For each loaded module, the Kernel generates device events. `udev` handles these events and generates the required special block files on a RAM file system in `/dev`. Without those special files, the file system and other devices would not be accessible.

Managing RAID and LVM Setups

If you configured your system to hold the root file system under RAID or LVM, `init` on `initramfs` sets up LVM or RAID to enable access to the root file system later.

In case you want to change your `/usr` or `swap` partitions directly without the help of YaST, further actions are needed. If you forget these steps, your system will start in emergency mode. To avoid starting in emergency mode, perform the following steps:

PROCEDURE 10.1: UPDATING INIT RAM DISK WHEN SWITCHING TO LOGICAL VOLUMES

1. Edit the corresponding entry in `/etc/fstab` and replace your previous partitions with the logical volume.

2. Execute the following commands:

```
root # mount -a
root # swapon -a
```

3. Regenerate your initial RAM disk (initramfs) with mkinitrd or dracut.
4. For z Systems, additionally run grub2-install.

Find more information about RAID and LVM in *Book “Deployment Guide”, Chapter 11 “Advanced Disk Setup”*.

Managing Network Configuration

If you configured your system to use a network-mounted root file system (mounted via NFS), init on initramfs must make sure that the proper network drivers are loaded and that they are set up to allow access to the root file system.

If the file system resides on a network block device like iSCSI or SAN, the connection to the storage server is also set up by init on initramfs. SUSE Linux Enterprise Server supports booting from a secondary iSCSI target if the primary target is not available. For more details regarding configuration of the booting iSCSI target refer to *Book “Storage Administration Guide”, Chapter 14 “Mass Storage over IP Networks: iSCSI”, Section 14.3.1 “Using YaST for the iSCSI Initiator Configuration”*.

When init on initramfs is called during the initial boot as part of the installation process, its tasks differ from those mentioned above:

Finding the Installation Medium

When starting the installation process, your machine loads an installation Kernel and a special init containing the YaST installer. The YaST installer is running in a RAM file system and needs to have information about the location of the installation medium to access it for installing the operating system.

Initiating Hardware Recognition and Loading Appropriate Kernel Modules

As mentioned in [Section 10.2, “initramfs”](#), the boot process starts with a minimum set of drivers that can be used with most hardware configurations. init starts an initial hardware scanning process that determines the set of drivers suitable for your hardware configuration. These drivers are used to generate a custom initramfs that is needed to boot the system. If the modules are not needed for boot but for coldplug, the modules can be loaded with systemd; for more information, see [Section 13.6.4, “Loading Kernel Modules”](#).

Loading the Installation System

When the hardware is properly recognized, the appropriate drivers are loaded. The udev program creates the special device files and init starts the installation system with the YaST installer.

Starting YaST

Finally, init starts YaST, which starts package installation and system configuration.

11 The Boot Loader GRUB 2

This chapter describes how to configure GRUB 2, the boot loader used in SUSE® Linux Enterprise Server. It is the successor of the traditional GRUB boot loader—now called “GRUB Legacy”. GRUB 2 has become the default boot loader in SUSE® Linux Enterprise Server since version 12. A YaST module is available for configuring the most important settings. The boot procedure as a whole is outlined in [Chapter 10, Booting a Linux System](#). For details on Secure Boot support for UEFI machines, see [Chapter 12, UEFI \(Unified Extensible Firmware Interface\)](#).

11.1 Main Differences between GRUB Legacy and GRUB 2

- The configuration is stored in different files.
- More file systems are supported (for example, Btrfs).
- Can directly read files stored on LVM or RAID devices.
- The user interface can be translated and altered with themes.
- Includes a mechanism for loading modules to support additional features, such as file systems, etc.
- Automatically searches for and generates boot entries for other kernels and operating systems, such as Windows.
- Includes a minimal Bash-like console.

11.2 Configuration File Structure

The configuration of GRUB 2 is based on the following files:

/boot/grub2/grub.cfg

This file contains the configuration of the GRUB 2 menu items. It replaces menu.lst used in GRUB Legacy. grub.cfg is automatically generated by the grub2-mkconfig command, and should not be edited.

/boot/grub2/custom.cfg

This optional file is directly sourced by grub.cfg at boot time and can be used to add custom items to the boot menu. Starting with SUSE Linux Enterprise Server these entries will also be parsed when using grub-once.

/etc/default/grub

This file controls the user settings of GRUB 2 and usually includes additional environmental settings such as backgrounds and themes.

Scripts under /etc/grub.d/

The scripts in this directory are read during execution of the grub2-mkconfig command. Their instructions are integrated into the main configuration file /boot/grub/grub.cfg.

/etc/sysconfig/bootloader

This configuration file is used when configuring the boot loader with YaST and every time a new kernel is installed. It is evaluated by the perl-bootloader which modifies the boot loader configuration file (for example /boot/grub2/grub.cfg for GRUB 2) accordingly. /etc/sysconfig/bootloader is not a GRUB 2-specific configuration file—the values are applied to any boot loader installed on SUSE Linux Enterprise Server.

/boot/grub2/x86_64-efi, /boot/grub2/power-ieee1275, /boot/grub2/s390x

These configuration files contain architecture-specific options.

GRUB 2 can be controlled in various ways. Boot entries from an existing configuration can be selected from the graphical menu (splash screen). The configuration is loaded from the file /boot/grub2/grub.cfg which is compiled from other configuration files (see below). All GRUB 2 configuration files are considered system files, and you need root privileges to edit them.



Note: Activating Configuration Changes

After having manually edited GRUB 2 configuration files, you need to run grub2-mkconfig to activate the changes. However, this is not necessary when changing the configuration with YaST, since it will automatically run grub2-mkconfig.

11.2.1 The File `/boot/grub2/grub.cfg`

The graphical splash screen with the boot menu is based on the GRUB 2 configuration file `/boot/grub2/grub.cfg`, which contains information about all partitions or operating systems that can be booted by the menu.

Every time the system is booted, GRUB 2 loads the menu file directly from the file system. For this reason, GRUB 2 does not need to be re-installed after changes to the configuration file. `grub.cfg` is automatically rebuilt with kernel installations or removals.

`grub.cfg` is compiled by the `grub2-mkconfig` from the file `/etc/default/grub` and scripts found in the `/etc/grub.d/` directory. Therefore you should never edit the file manually. Instead, edit the related source files or use the YaST *Boot Loader* module to modify the configuration as described in [Section 11.3, “Configuring the Boot Loader with YaST”](#).

11.2.2 The File `/etc/default/grub`

More general options of GRUB 2 belong here, such as the time the menu is displayed, or the default OS to boot. To list all available options, see the output of the following command:

```
grep "export GRUB_DEFAULT" -A50 /usr/sbin/grub2-mkconfig | grep GRUB_
```

In addition to already defined variables, the user may introduce their own variables, and use them later in the scripts found in the `/etc/grub.d` directory.

After having edited `/etc/default/grub`, run `grub2-mkconfig` to update the main configuration file.



Note: Scope

All options set in this file are general options that affect all boot entries. Specific options for Xen Kernels or the Xen hypervisor can be set via the `GRUB_*_XEN_*` configuration options. See below for details.

GRUB_DEFAULT

Sets the boot menu entry that is booted by default. Its value can be a numeric value, the complete name of a menu entry, or “saved”.

`GRUB_DEFAULT=2` boots the third (counted from zero) boot menu entry.

`GRUB_DEFAULT="2>0"` boots the first submenu entry of the third top-level menu entry.

GRUB_DEFAULT="Example boot menu entry" boots the menu entry with the title “Example boot menu entry”.

GRUB_DEFAULT=saved boots the entry specified by the grub2-reboot or grub2-set-default commands. While grub2-reboot sets the default boot entry for the next reboot only, grub2-set-default sets the default boot entry until changed.

GRUB_HIDDEN_TIMEOUT

Waits the specified number of seconds for the user to press a key. During the period no menu is shown unless the user presses a key. If no key is pressed during the time specified, the control is passed to GRUB_TIMEOUT. GRUB_HIDDEN_TIMEOUT=0 first checks whether Shift is pressed and shows the boot menu if yes, otherwise immediately boots the default menu entry. This is the default when only one bootable OS is identified by GRUB 2.

GRUB_HIDDEN_TIMEOUT_QUIET

If false is specified, a countdown timer is displayed on a blank screen when the GRUB_HIDDEN_TIMEOUT feature is active.

GRUB_TIMEOUT

Time period in seconds the boot menu is displayed before automatically booting the default boot entry. If you press a key, the timeout is cancelled and GRUB 2 waits for you to make the selection manually. GRUB_TIMEOUT=-1 will cause the menu to be displayed until you select the boot entry manually.

GRUB_CMDLINE_LINUX

Entries on this line are added at the end of the boot entries for normal and recovery mode. Use it to add kernel parameters to the boot entry.

GRUB_CMDLINE_LINUX_DEFAULT

Same as GRUB_CMDLINE_LINUX but the entries are appended in the normal mode only.

GRUB_CMDLINE_LINUX_RECOVERY

Same as GRUB_CMDLINE_LINUX but the entries are appended in the recovery mode only.

GRUB_CMDLINE_LINUX_XEN_REPLACE

This entry will completely replace the GRUB_CMDLINE_LINUX parameters for all Xen boot entries.

GRUB_CMDLINE_LINUX_XEN_REPLACE_DEFAULT

Same as GRUB_CMDLINE_LINUX_XEN_REPLACE but it will only replace parameters of GRUB_CMDLINE_LINUX_DEFAULT.

GRUB_CMDLINE_XEN

This entry specifies the kernel parameters for the Xen guest kernel only—the operation principle is the same as for GRUB_CMDLINE_LINUX.


GRUB_CMDLINE_XEN_DEFAULT

Same as GRUB_CMDLINE_XEN—the operation principle is the same as for GRUB_CMDLINE_LINUX_DEFAULT.

GRUB_TERMINAL

Enables and specifies an input/output terminal device. Can be console (PC BIOS and EFI consoles), serial (serial terminal), ofconsole (Open Firmware console), or the default gfxterm (graphics-mode output). It is also possible to enable more than one device by quoting the required options, for example GRUB_TERMINAL="console serial".

GRUB_GFXMODE

The resolution used for the gfxterm graphical terminal. Note that you can only use modes supported by your graphics card (VBE). The default is ‘auto’, which tries to select a preferred resolution. You can display the screen resolutions available to GRUB 2 by typing vbeinfo in the GRUB 2 command line. The command line is accessed by typing  when the GRUB 2 boot menu screen is displayed.

You can also specify a color depth by appending it to the resolution setting, for example GRUB_GFXMODE=1280x1024x24.

GRUB_BACKGROUND

Set a background image for the gfxterm graphical terminal. The image must be a file readable by GRUB 2 at boot time, and it must end with the .png, .tga, .jpg, or .jpeg suffix. If necessary, the image will be scaled to fit the screen.

GRUB_DISABLE_OS_PROBER

If this option is set to true, automatic searching for other operating systems is disabled. Only the kernel images in /boot/ and the options from your own scripts in /etc/grub.d/ are detected.

SUSE_BTRFS_SNAPSHOT_BOOTING

If this option is set to true, GRUB 2 can boot directly into Snapper snapshots. For more information, see [Section 6.3, “System Rollback by Booting from Snapshots”](#).



Note: Parameter Handling

All *_DEFAULT parameters can be configured manually or with YaST.

For a complete list of options, see the [GNU GRUB manual \(http://www.gnu.org/software/grub/manual/grub.html#Simple-configuration\)](http://www.gnu.org/software/grub/manual/grub.html#Simple-configuration). For a complete list of possible parameters, see <http://en.opensuse.org/Linuxrc>.

11.2.3 Scripts in `/etc/grub.d`

The scripts in this directory are read during execution of the `grub2-mkconfig` command, and their instructions are incorporated into `/boot/grub2/grub.cfg`. The order of menu items in `grub.cfg` is determined by the order in which the files in this directory are run. Files with a leading numeral are executed first, beginning with the lowest number. `00_header` is run before `10_linux`, which would run before `40_custom`. If files with alphabetic names are present, they are executed after the numerically-named files. Only executable files generate output to `grub.cfg` during execution of `grub2-mkconfig`. By default all files in the `/etc/grub.d` directory are executable. The most important scripts are:

00_header

Sets environmental variables such as system file locations, display settings, themes, and previously saved entries. It also imports preferences stored in the `/etc/default/grub`. Normally you do not need to make changes to this file.

10_linux

Identifies Linux kernels on the root device and creates relevant menu entries. This includes the associated recovery mode option if enabled. Only the latest kernel is displayed on the main menu page, with additional kernels included in a submenu.

30_os-prober

This script uses `OS-prober` to search for Linux and other operating systems and places the results in the GRUB 2 menu. There are sections to identify specific other operating systems, such as Windows or macOS.

40_custom

This file provides a simple way to include custom boot entries into `grub.cfg`. Make sure that you do not change the `exec tail -n +3 $0` part at the beginning.

90_persistent

This is a special script that copies a corresponding part of the `grub.cfg` file and outputs it back unchanged. This way you can modify that part of `grub.cfg` directly and the change survives the execution of `grub2-mkconfig`.

The processing sequence is set by the preceding numbers with the lowest number being executed first. If scripts are preceded by the same number the alphabetical order of the complete name decides the order.

11.2.4 Mapping between BIOS Drives and Linux Devices

In GRUB Legacy, the `device.map` configuration file was used to derive Linux device names from BIOS drive numbers. The mapping between BIOS drives and Linux devices cannot always be guessed correctly. For example, GRUB Legacy would get a wrong order if the boot sequence of IDE and SCSI drives is exchanged in the BIOS configuration.

GRUB 2 avoids this problem by using device ID strings (UUIDs) or file system labels when generating `grub.cfg`. GRUB 2 utilities create a temporary device map on the fly, which is usually sufficient, particularly in the case of single-disk systems.

However, if you need to override the GRUB 2's automatic device mapping mechanism, create your custom mapping file `/boot/grub2/device.map`. The following example changes the mapping to make `DISK 3` the boot disk. Note that GRUB 2 partition numbers start with `1` and not with `0` as in GRUB Legacy.

```
(hd1) /dev/disk-by-id/DISK3 ID
(hd2) /dev/disk-by-id/DISK1 ID
(hd3) /dev/disk-by-id/DISK2 ID
```

11.2.5 Editing Menu Entries during the Boot Procedure

Being able to directly edit menu entries is useful when the system does not boot anymore because of a faulty configuration. It can also be used to test new settings without altering the system configuration.

1. In the graphical boot menu, select the entry you want to edit with the arrow keys.
2. Press `E` to open the text-based editor.
3. Use the arrow keys to move to the line you want to edit.

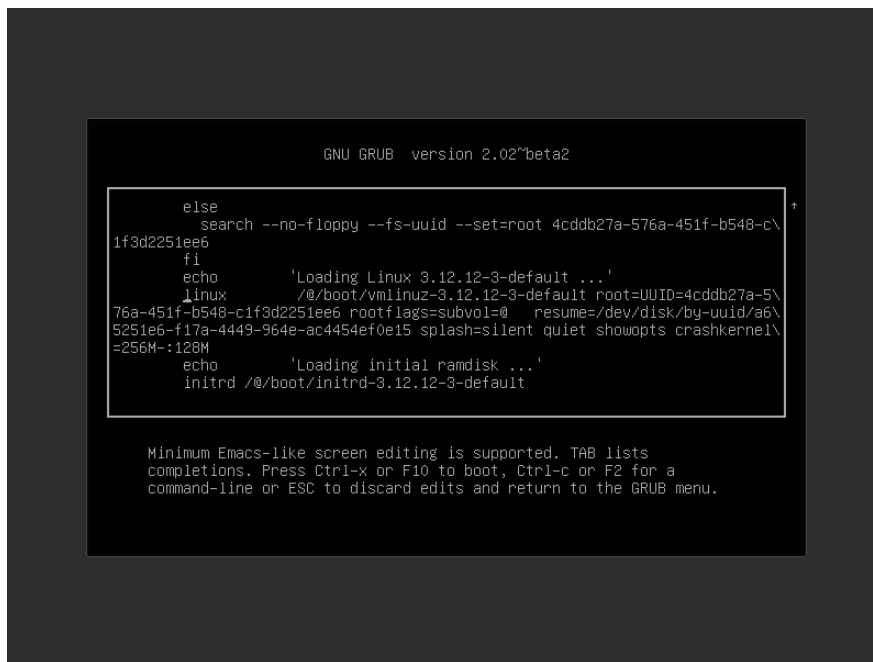


FIGURE 11.1: GRUB 2 BOOT EDITOR

Now you have two options:

- a. Add space-separated parameters to the end of the line starting with `linux` or `linuxefi` to edit the kernel parameters. A complete list of parameters is available at <http://en.opensuse.org/Linuxrc>.
 - b. Or edit the general options to change for example the kernel version. The `→|` key suggests all possible completions.
4. Press `F10` to boot the system with the changes you made or press `Esc` to discard your edits and return to the GRUB 2 menu.

Changes made this way only apply to the current boot process and are not saved permanently.

! Important: Keyboard Layout During the Boot Procedure

The US keyboard layout is the only one available when booting. See *Figure 38.2, "US Keyboard Layout"*.



Note: Boot Loader on the Installation Media

The Boot Loader of the installation media on systems with a traditional BIOS is still GRUB Legacy. To add boot options, select an entry and start typing. Additions you make to the installation boot entry will be permanently saved in the installed system.



Note: Editing GRUB 2 Menu Entries on z Systems

Cursor movement and editing commands on IBM z Systems differ—see [Section 11.4, “Differences in Terminal Usage on z Systems”](#) for details.

11.2.6 Setting a Boot Password

Even before the operating system is booted, GRUB 2 enables access to file systems. Users without root permissions can access files in your Linux system to which they have no access after the system is booted. To block this kind of access or to prevent users from booting certain menu entries, set a boot password.



Important: Booting Requires Password

If set, the boot password is required on every boot, which means the system does not boot automatically.

Proceed as follows to set a boot password. Alternatively use YaST ([Protect Boot Loader with Password](#)).

1. Encrypt the password using **`grub2-mkpasswd-pbkdf2`**:

```
tux > sudo grub2-mkpasswd-pbkdf2
Password: ****
Reenter password: ****
PBKDF2 hash of your password is grub.pbkdf2.sha512.10000.9CA4611006FE96BC77A...
```

2. Paste the resulting string into the file `/etc/grub.d/40_custom` together with the **`set superusers`** command.

```
set superusers="root"
```



```
password_pbkdf2 root grub.pbkdf2.sha512.10000.9CA4611006FE96BC77A...
```

3. Run **grub2-mkconfig** to import the changes into the main configuration file.

After you reboot, you will be prompted for a user name and a password when trying to boot a menu entry. Enter `root` and the password you typed during the **grub2-mkpasswd-pbkdf2** command. If the credentials are correct, the system will boot the selected boot entry.

For more information, see <https://www.gnu.org/software/grub/manual/grub.html#Security>.

11.3 Configuring the Boot Loader with YaST

The easiest way to configure general options of the boot loader in your SUSE Linux Enterprise Server system is to use the YaST module. In the *YaST Control Center*, select *System > Boot Loader*. The module shows the current boot loader configuration of your system and allows you to make changes.

Use the *Boot Code Options* tab to view and change settings related to type, location and advanced loader settings. You can choose whether to use GRUB 2 in standard or EFI mode.

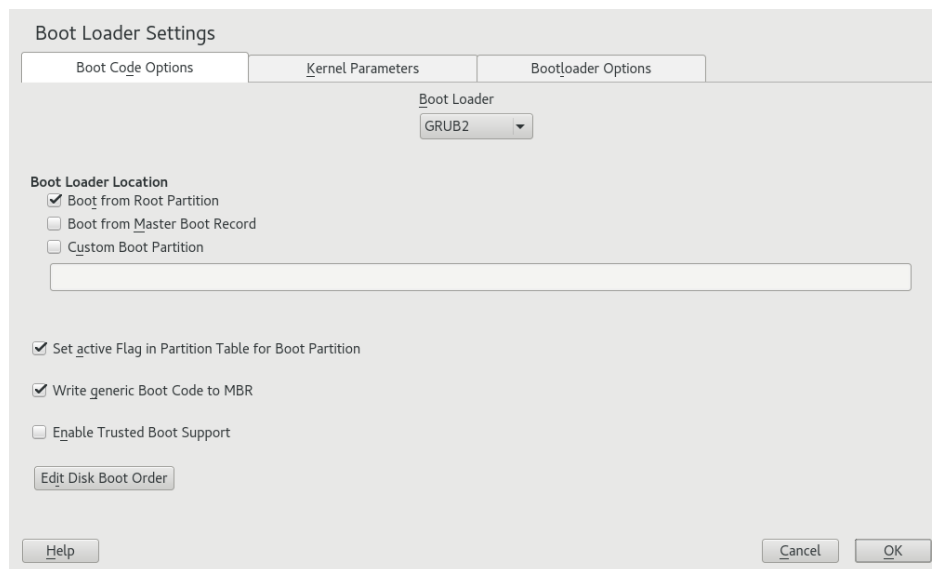


FIGURE 11.2: BOOT CODE OPTIONS

! Important: EFI Systems require GRUB2-EFI

If you have an EFI system you can only install GRUB2-EFI, otherwise your system is no longer bootable.

Important: Reinstalling the Boot Loader

To reinstall the boot loader, make sure to change a setting in YaST and then change it back. For example, to reinstall GRUB2-EFI, select *GRUB2* first and then immediately switch back to *GRUB2-EFI*.

Otherwise, the boot loader may only be partially reinstalled.

Note: Custom Boot Loader

To use a boot loader other than the ones listed, select *Do Not Install Any Boot Loader*. Read the documentation of your boot loader carefully before choosing this option.

11.3.1 Modifying the Boot Loader Location

To modify the location of the boot loader, follow these steps:

PROCEDURE 11.1: CHANGING THE BOOT LOADER LOCATION

1. Select the *Boot Code Options* tab and then choose one of the following options for *Boot Loader Location*:

Boot from Master Boot Record

This installs the boot loader in the MBR of the first disk (according to the boot sequence preset in the BIOS).

Boot from Root Partition

This installs the boot loader in the boot sector of the partition (this is the default).

Custom Boot Partition

Use this option to specify the location of the boot loader manually.

2. Click *OK* to apply your changes.

11.3.2 Adjusting the Disk Order

If your computer has more than one hard disk, you can specify the boot sequence of the disks. For more information, see [Section 11.2.4, “Mapping between BIOS Drives and Linux Devices”](#).

PROCEDURE 11.2: SETTING THE DISK ORDER

1. Open the *Boot Code Options* tab.
2. Click *Boot Loader Installation Details*.
3. If more than one disk is listed, select a disk and click *Up* or *Down* to reorder the displayed disks.
4. Click *OK* two times to save the changes.

11.3.3 Configuring Advanced Options

Advanced boot options can be configured via the *Boot Loader Options* tab.

11.3.3.1 *Boot Loader Options* Tab

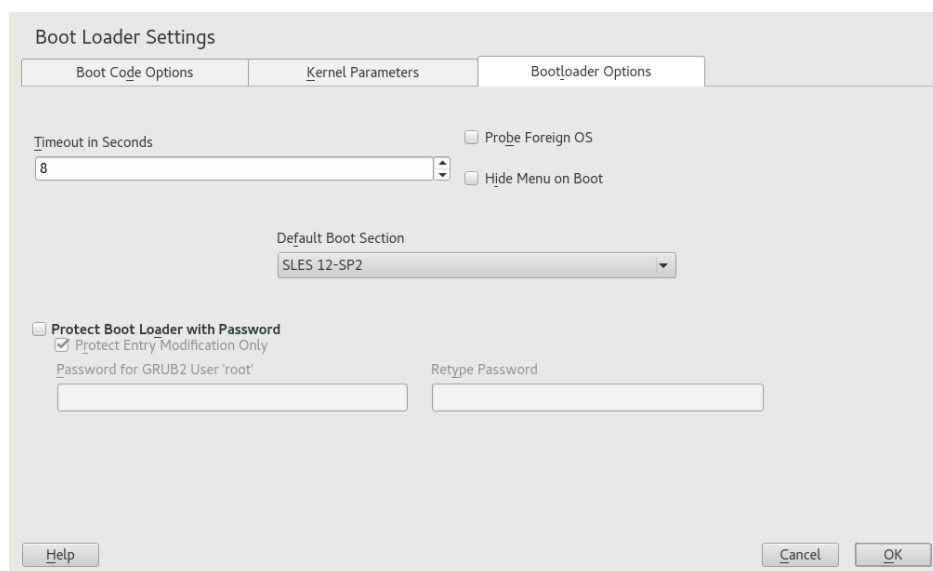


FIGURE 11.3: BOOT LOADER OPTIONS

Boot Loader Time-Out

Change the value of *Time-Out in Seconds* by typing in a new value and clicking the appropriate arrow key with your mouse.

Probe Foreign OS

When selected, the boot loader searches for other systems like Windows or other Linux installations.

Hide Menu on Boot

Hides the boot menu and boots the default entry.

Adjusting the Default Boot Entry

Select the desired entry from the “Default Boot Section” list. Note that the “>” sign in the boot entry name delimits the boot section and its subsection.

Protect Boot Loader with Password

Protects the boot loader and the system with an additional password. For more information, see [Section 11.2.6, “Setting a Boot Password”](#).

11.3.3.2 Kernel Parameters Tab

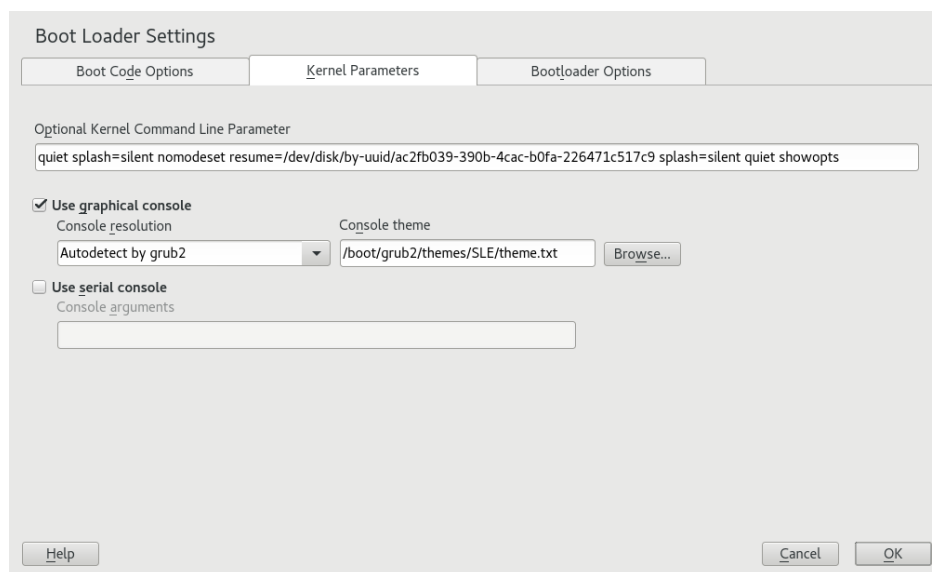


FIGURE 11.4: KERNEL PARAMETERS

VGA Mode

The VGA Mode option specifies the default screen resolution during the boot process.

Kernel Command Line Parameter

The optional kernel parameters are added at the end of the default parameters. For a list of all possible parameters, see <http://en.opensuse.org/Linuxrc>.

Use graphical console

When checked, the boot menu appears on a graphical splash screen rather than in a text mode. The resolution of the boot screen can be then set from the *Console resolution* list, and graphical theme definition file can be specified with the *Console theme* file-chooser.

Use Serial Console

If your machine is controlled via a serial console, activate this option and specify which COM port to use at which speed. See [info grub](#) or <http://www.gnu.org/software/grub/manual/grub.html#Serial-terminal> ↗

11.3.3.3 Boot Code Options Tab

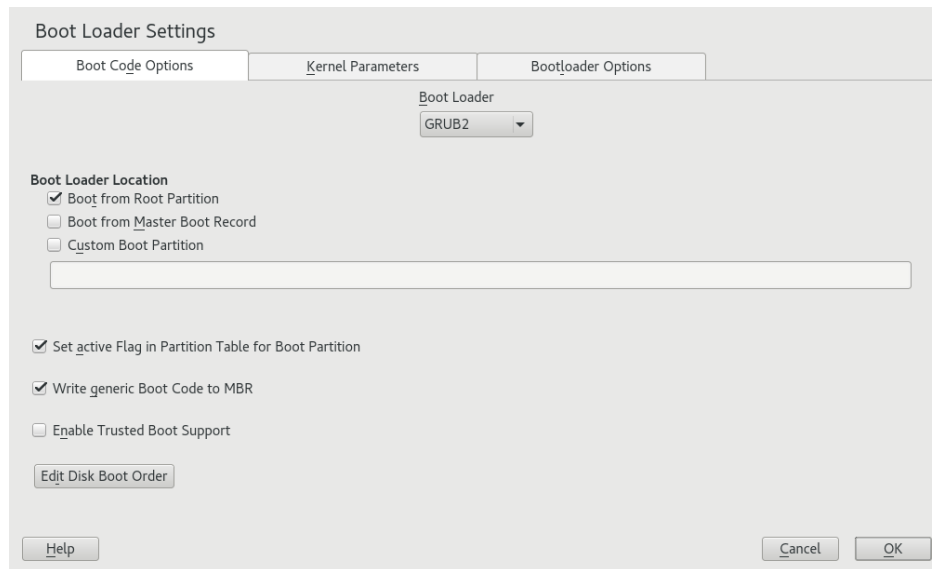


FIGURE 11.5: CODE OPTIONS

Set Active Flag in Partition Table for Boot Partition

Activates the partition that contains the boot loader. Some legacy operating systems (such as Windows) can only boot from an active partition.

Write Generic Boot Code to MBR

Replaces the current MBR with generic, operating system independent code.

Enable Trusted Boot Support

Starts TrustedGRUB2 which supports trusted computing functionality (Trusted Platform Module (TPM)). For more information refer to <https://github.com/Sirrix-AG/Trusted-GRUB2> ↗.

11.4 Differences in Terminal Usage on z Systems

On 3215 and 3270 terminals there are some differences and limitations on how to move the cursor and how to issue editing commands within GRUB 2.

11.4.1 Limitations

Interactivity

Interactivity is strongly limited. Typing often does not result in visual feedback. To see where the cursor is, type an underscore (`_`).



Note: 3270 Compared to 3215

The 3270 terminal is much better at displaying and refreshing screens than the 3215 terminal.

Cursor Movement

“Traditional” cursor movement is not possible. `Alt`, `Meta`, `Ctrl` and the cursor keys do not work. To move the cursor, use the key combinations listed in [Section 11.4.2, “Key Combinations”](#).

Caret




















The caret (`^`) is used as a control character. To type a literal `^` followed by a letter, type `^`, `^`, LETTER.
















Enter

The `Enter` key does not work, use `^J` instead.

11.4.2 Key Combinations

Common Substitutes:	<code>^J</code>	engage (“Enter”)
	<code>^L</code>	abort, return to previous “state”

	 -I	tab completion (in edit and shell mode)
Keys Available in Menu Mode:	 -A	first entry
	 -E	last entry
	 -P	previous entry
	 -N	next entry
	 -G	previous page
	 -C	next page
	 -F	boot selected entry or enter submenu (same as  -J)
	E	edit selected entry
	C	enter GRUB-Shell
Keys Available in Edit Mode:	 -P	previous line
	 -N	next line
	 -B	backward char
	 -F	forward char
	 -A	beginning of line
	 -E	end of line
	 -H	backspace
	 -D	delete
	 -K	kill line
	 -Y	yank

	 -O	open line
	 -L	refresh screen
	 -X	boot entry
	 -C	enter GRUB-Shell
Keys Available in Command Line Mode:	 -P	previous command
	 -N	next command from history
	 -A	beginning of line
	 -E	end of line
	 -B	backward char
	 -F	forward char
	 -H	backspace
	 -D	delete
	 -K	kill line
	 -U	discard line
	 -Y	yank

11.5 Helpful GRUB 2 Commands

grub2-mkconfig

Generates a new /boot/grub2/grub.cfg based on /etc/default/grub and the scripts from /etc/grub.d/.

EXAMPLE 11.1: USAGE OF GRUB2-MKCONFIG

```
grub2-mkconfig -o /boot/grub2/grub.cfg
```




Tip: Syntax Check

Running **grub2-mkconfig** without any parameters prints the configuration to STD-OUT where it can be reviewed. Use **grub2-script-check** after `/boot/grub2/grub.cfg` has been written to check its syntax.



Important: **grub2-mkconfig** Cannot Repair UEFI Secure Boot Tables

If you are using UEFI Secure Boot and your system is not reaching GRUB 2 correctly anymore, you may need to additionally reinstall Shim and regenerate the UEFI boot table. To do so, use:

```
root # shim-install --config-file=/boot/grub2/grub.cfg
```

grub2-mkrescue

Creates a bootable rescue image of your installed GRUB 2 configuration.

EXAMPLE 11.2: USAGE OF GRUB2-MKRESCUE

```
grub2-mkrescue -o save_path/name.iso iso
```

grub2-script-check

Checks the given file for syntax errors.

EXAMPLE 11.3: USAGE OF GRUB2-SCRIPT-CHECK

```
grub2-check-config /boot/grub2/grub.cfg
```

grub2-once

Set the default boot entry for the next boot only. To get the list of available boot entries use the `--list` option.

EXAMPLE 11.4: USAGE OF GRUB2-ONCE



```
grub2-once number_of_the_boot_entry
```



Tip: **grub2-once** Help

Call the program without any option to get a full list of all possible options.

11.6 More Information

Extensive information about GRUB 2 is available at <http://www.gnu.org/software/grub/> . Also refer to the [grub](#) info page. You can also search for the keyword “GRUB 2” in the Technical Information Search at <http://www.suse.com/support>  to get information about special issues.

12 UEFI (Unified Extensible Firmware Interface)

UEFI (Unified Extensible Firmware Interface) is the interface between the firmware that comes with the system hardware, all the hardware components of the system, and the operating system. UEFI is becoming more and more available on PC systems and thus is replacing the traditional PC-BIOS. UEFI, for example, properly supports 64-bit systems and offers secure booting (“Secure Boot”, firmware version 2.3.1c or better required), which is one of its most important features. Lastly, with UEFI a standard firmware will become available on all x86 platforms.

UEFI additionally offers the following advantages:

- Booting from large disks (over 2 TiB) with a GUID Partition Table (GPT).
- CPU-independent architecture and drivers.
- Flexible pre-OS environment with network capabilities.
- CSM (Compatibility Support Module) to support booting legacy operating systems via a PC-BIOS-like emulation.

For more information, see http://en.wikipedia.org/wiki/Unified_Extensible_Firmware_Interface. The following sections are not meant as a general UEFI overview; these are only hints about how some features are implemented in SUSE Linux Enterprise.

12.1 Secure Boot

In the world of UEFI, securing the bootstrapping process means establishing a chain of trust. The “platform” is the root of this chain of trust; in the context of SUSE Linux Enterprise, the mainboard and the on-board firmware could be considered the “platform”. Or, put slightly differently, it is the hardware vendor, and the chain of trust flows from that hardware vendor to the component manufacturers, the OS vendors, etc.

The trust is expressed via public key cryptography. The hardware vendor puts a so-called Platform Key (PK) into the firmware, representing the root of trust. The trust relationship with operating system vendors and others is documented by signing their keys with the Platform Key. Finally, security is established by requiring that no code will be executed by the firmware unless it has been signed by one of these “trusted” keys—be it an OS boot loader, some driver located in the flash memory of some PCI Express card or on disk, or be it an update of the firmware itself.

Essentially, to use Secure Boot, you need to have your OS loader signed with a key trusted by the firmware, and you need the OS loader to verify that the kernel it loads can be trusted.

Key Exchange Keys (KEK) can be added to the UEFI key database. This way, you can use other certificates, as long as they are signed with the private part of the PK.

12.1.1 Implementation on SUSE Linux Enterprise

Microsoft's Key Exchange Key (KEK) is installed by default.



Note: GUID Partitioning Table (GPT) Required

The Secure Boot feature is enabled by default on UEFI/x86_64 installations. You can find the *Enable Secure Boot Support* option in the *Boot Code Options* tab of the *Boot Loader Settings* dialog. It supports booting when the secure boot is activated in the firmware, while making it possible to boot when it is deactivated.

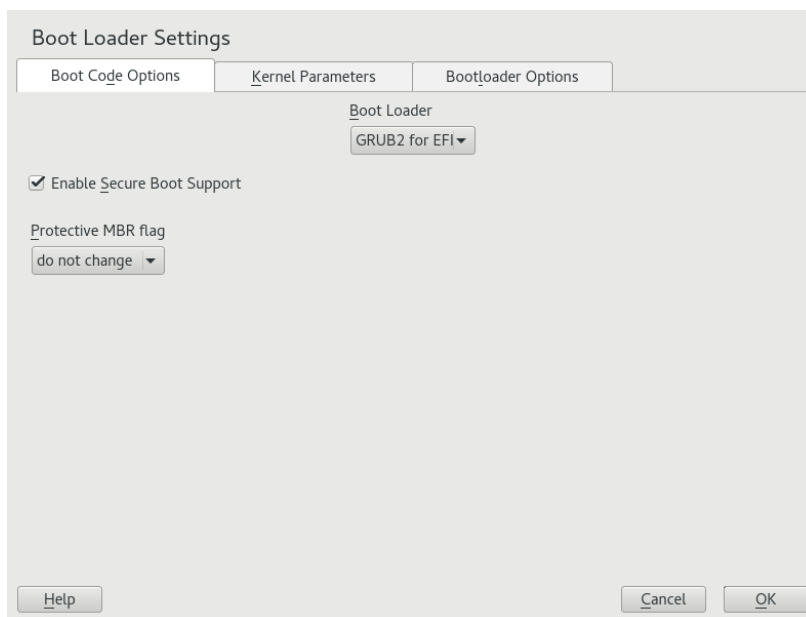


FIGURE 12.1: **SECURE BOOT SUPPORT**

The Secure Boot feature requires that a GUID Partitioning Table (GPT) replaces the old partitioning with a Master Boot Record (MBR). If YaST detects EFI mode during the installation, it will try to create a GPT partition. UEFI expects to find the EFI programs on a FAT-formatted EFI System Partition (ESP).

Supporting UEFI Secure Boot essentially requires having a boot loader with a digital signature that the firmware recognizes as a trusted key. To be useful for SUSE Linux Enterprise customers, that key is trusted by the firmware a priori, without requiring any manual intervention.

There are two ways of getting there. One is to work with hardware vendors to have them endorse a SUSE key, which SUSE then signs the boot loader with. The other way is to go through Microsoft's Windows Logo Certification program to have the boot loader certified and have Microsoft recognize the SUSE signing key (that is, have it signed with their KEK). By now, SUSE got the loader signed by UEFI Signing Service (that is Microsoft in this case).

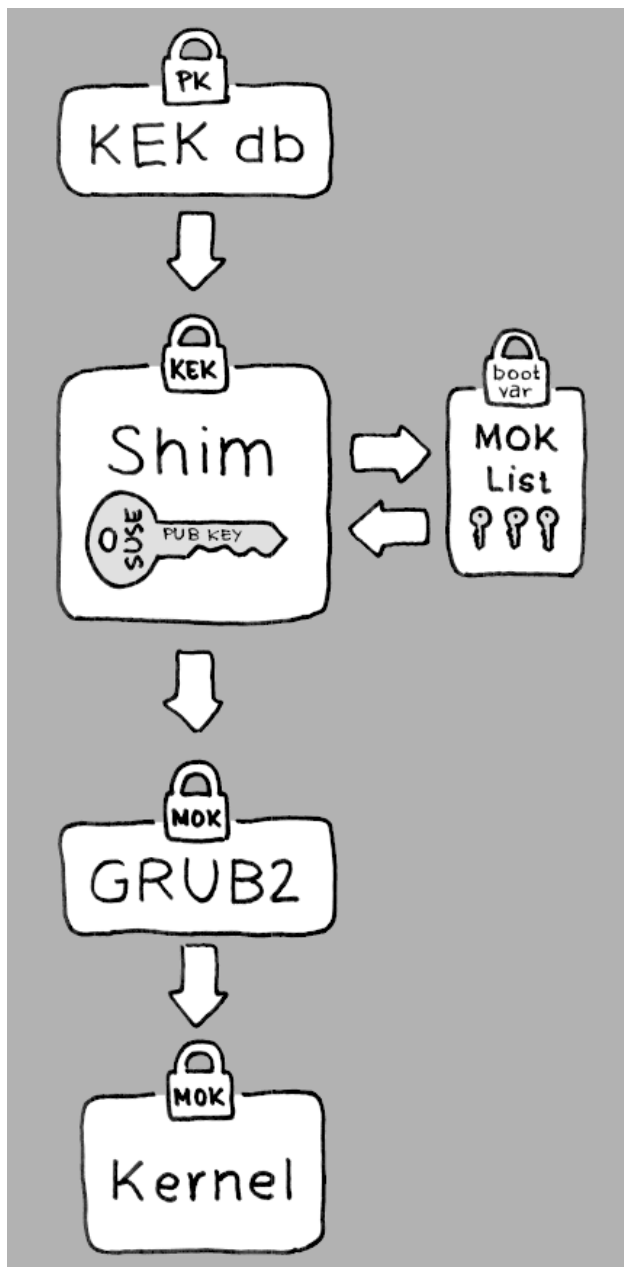


FIGURE 12.2: **UEFI: SECURE BOOT PROCESS**

At the implementation layer, SUSE uses the shim loader which is installed by default. It is a smart solution that avoids legal issues, and simplifies the certification and signing step considerably. The shim loader's job is to load a boot loader such as GRUB 2 and verify it; this boot loader in turn will load kernels signed by a SUSE key only. SUSE provides this functionality since SLE11 SP3 on fresh installations with UEFI Secure Boot enabled.

There are two types of trusted users:

- First, those who hold the keys. The Platform Key (PK) allows almost everything. The Key Exchange Key (KEK) allows all a PK can except changing the PK.
- Second, anyone with physical access to the machine. A user with physical access can reboot the machine, and configure UEFI.

UEFI offers two types of variables to fulfill the needs of those users:

- The first is the so-called “Authenticated Variables”, which can be updated from both within the boot process (the so-called Boot Services Environment) and the running OS, but only when the new value of the variable is signed with the same key that the old value of the variable was signed with. And they can only be appended to or changed to a value with a higher serial number.
- The second is the so-called “Boot Services Only Variables”. These variables are accessible to any code that runs during the boot process. After the boot process ends and before the OS starts, the boot loader must call the `ExitBootServices` call. After that, these variables are no longer accessible, and the OS cannot touch them.

The various UEFI key lists are of the first type, as this allows online updating, adding, and blacklisting of keys, drivers, and firmware fingerprints. It is the second type of variable, the “Boot Services Only Variable”, that helps to implement Secure Boot, in a matter that is both secure and open source friendly, and thus compatible with GPLv3.

SUSE starts with `shim`—a small and simple EFI boot loader—which was originally developed by Fedora. It is signed by a certificate signed by the SUSE KEK and a Microsoft-issued certificate, based on which KEKs are available in the UEFI key database on the system.

This allows `shim` to load and execute.

`shim` then goes on to verify that the boot loader it wants to load is trusted. In a default situation `shim` will use an independent SUSE certificate embedded in its body. In addition, `shim` will allow to “enroll” additional keys, overriding the default SUSE key. In the following, we call them “Machine Owner Keys” or MOKs for short.

Next the boot loader will verify and then boot the kernel, and the kernel will do the same on the modules.

12.1.2 MOK (Machine Owner Key)

If the user (“machine owner”) wants to replace any components of the boot process, Machine Owner Keys (MOKs) are to be used. The `mokutils` tool will help with signing components and managing MOKs.

The enrollment process begins with rebooting the machine and interrupting the boot process (for example, pressing a key) when `shim` loads. `shim` will then go into enrollment mode, allowing the user to replace the default SUSE key with keys from a file on the boot partition. If the user chooses to do so, `shim` will then calculate a hash of that file and put the result in a “Boot Services Only” variable. This allows `shim` to detect any change of the file made outside of Boot Services and thus avoid tampering with the list of user-approved MOKs.

All of this happens during boot time—only verified code is executing now. Therefore, only a user present at the console can use the machine owner's set of keys. It cannot be malware or a hacker with remote access to the OS because hackers or malware can only change the file, but not the hash stored in the “Boot Services Only” variable.

The boot loader, after having been loaded and verified by `shim`, will call back to `shim` when it wants to verify the kernel—to avoid duplication of the verification code. `Shim` will use the same list of MOKs for this and tell the boot loader whether it can load the kernel.

This way, you can install your own kernel or boot loader. It is only necessary to install a new set of keys and authorize them by being physically present during the first reboot. Because MOKs are a list and not just a single MOK, you can make `shim` trust keys from several vendors, allowing dual- and multi-boot from the boot loader.

12.1.3 Booting a Custom Kernel

The following is based on http://en.opensuse.org/openSUSE:UEFI#Booting_a_custom_kernel.

Secure Boot does not prevent you from using a self-compiled kernel. You must sign it with your own certificate and make that certificate known to the firmware or MOK.

1. Create a custom X.509 key and certificate used for signing:

```
openssl req -new -x509 -newkey rsa:2048 -keyout key.asc \
-out cert.pem -nodes -days 666 -subj "/CN=$USER/"
```

For more information about creating certificates, see http://en.opensuse.org/openSUSE:UEFI_Image_File_Sign_Tools#Create_Your_Own_Certificate.

2. Package the key and the certificate as a PKCS#12 structure:

```
openssl pkcs12 -export -inkey key.asc -in cert.pem \  
-name kernel_cert -out cert.p12
```

3. Generate an NSS database for use with **pesign**:

```
certutil -d . -N
```

4. Import the key and the certificate contained in PKCS#12 into the NSS database:

```
pk12util -d . -i cert.p12
```

5. “Bless” the kernel with the new signature using **pesign**:

```
pesign -n . -c kernel_cert -i arch/x86/boot/bzImage \  
-o vmlinuz.signed -s
```

6. List the signatures on the kernel image:

```
pesign -n . -S -i vmlinuz.signed
```

At that point, you can install the kernel in /boot as usual. Because the kernel now has a custom signature the certificate used for signing needs to be imported into the UEFI firmware or MOK.

7. Convert the certificate to the DER format for import into the firmware or MOK:

```
openssl x509 -in cert.pem -outform der -out cert.der
```

8. Copy the certificate to the ESP for easier access:

```
sudo cp cert.der /boot/efi/
```

9. Use **mokutil** to launch the MOK list automatically.

- **a.** Import the certificate to MOK:

```
mokutil --root-pw --import cert.der
```

The --root-pw option enables usage of the root user directly.

- b. Check the list of certificates that are prepared to be enrolled:

```
mokutil --list-new
```

- c. Reboot the system; shim should launch MokManager. You need to enter the root password to confirm the import of the certificate to the MOK list.

- d. Check if the newly imported key was enrolled:

```
mokutil --list-enrolled
```

- a. Alternatively, this is the procedure if you want to launch MOK manually:
Reboot

- b. In the GRUB 2 menu press the 'c' key.

- c. Type:

```
chainloader $efibootdir/MokManager.efi  
boot
```

- d. Select *Enroll key from disk*.

- e. Navigate to the cert.der file and press .

- f. Follow the instructions to enroll the key. Normally this should be pressing '0' and then 'y' to confirm.

Alternatively, the firmware menu may provide ways to add a new key to the Signature Database.

12.1.4 Using Non-Inbox Drivers

There is no support for adding non-inbox drivers (that is, drivers that do not come with SLE) during installation with Secure Boot enabled. The signing key used for SolidDriver/PLDP is not trusted by default.

It is possible to install third party drivers during installation with Secure Boot enabled in two different ways. In both cases:

- Add the needed keys to the firmware database via firmware/system management tools before the installation. This option depends on the specific hardware you are using. Consult your hardware vendor for more information.
- Use a bootable driver ISO from <https://drivers.suse.com/> or your hardware vendor to enroll the needed keys in the MOK list at first boot.

To use the bootable driver ISO to enroll the driver keys to the MOK list, follow these steps:

1. Burn the ISO image above to an empty CD/DVD medium.
2. Start the installation using the new CD/DVD medium, having the standard SUSE Linux Enterprise media at hand or a URL to a network installation server.
If doing a network installation, enter the URL of the network installation source on the boot command line using the `install=` option.
If doing installation from optical media, the installer will first boot from the driver kit and then ask to insert the first disk of the SUSE Linux Enterprise product.
3. An initrd containing updated drivers will be used for installation.

For more information, see https://drivers.suse.com/doc/Usage/Secure_Boot_Certificate.html.

12.1.5 Features and Limitations

When booting in Secure Boot mode, the following features apply:

- Installation to UEFI default boot loader location, a mechanism to keep or restore the EFI boot entry.
- Reboot via UEFI.
- Xen hypervisor will boot with UEFI when there is no legacy BIOS to fall back to.
- UEFI IPv6 PXE boot support.
- UEFI videomode support, the kernel can retrieve video mode from UEFI to configure KMS mode with the same parameters.
- UEFI booting from USB devices is supported.

When booting in Secure Boot mode, the following limitations apply:


- To ensure that Secure Boot cannot be easily circumvented, some kernel features are disabled when running under Secure Boot.
- Boot loader, kernel, and kernel modules must be signed.
- Kexec and Kdump are disabled.
- Hibernation (suspend on disk) is disabled.
- Access to `/dev/kmem` and `/dev/mem` is not possible, not even as root user.
- Access to the I/O port is not possible, not even as root user. All X11 graphical drivers must use a kernel driver.
- PCI BAR access through sysfs is not possible.
- `custom_method` in ACPI is not available.
- debugfs for asus-wmi module is not available.
- the `acpi_rsdp` parameter does not have any effect on the kernel.

12.2 For More Information

- <http://www.uefi.org> —UEFI home page where you can find the current UEFI specifications.
- Blog posts by Olaf Kirch and Vojtěch Pavlík (the chapter above is heavily based on these posts):
 - <http://www.suse.com/blogs/uefi-secure-boot-plan/>
 - <http://www.suse.com/blogs/uefi-secure-boot-overview/>
 - <http://www.suse.com/blogs/uefi-secure-boot-details/>
- <http://en.opensuse.org/openSUSE:UEFI> —UEFI with openSUSE.

13 The systemd Daemon

The program `systemd` is the process with process ID 1. It is responsible for initializing the system in the required way. `systemd` is started directly by the Kernel and resists signal 9, which normally terminates processes. All other programs are either started directly by `systemd` or by one of its child processes.

Starting with SUSE Linux Enterprise Server 12 `systemd` is a replacement for the popular System V init daemon. `systemd` is fully compatible with System V init (by supporting init scripts). One of the main advantages of `systemd` is that it considerably speeds up boot time by aggressively paralleling service starts. Furthermore, `systemd` only starts a service when it is really needed. Daemons are not started unconditionally at boot time, but rather when being required for the first time. `systemd` also supports Kernel Control Groups (cgroups), snapshotting and restoring the system state and more. See <http://www.freedesktop.org/wiki/Software/systemd/>  for details.

13.1 The systemd Concept

This section will go into detail about the concept behind `systemd`.

13.1.1 What Is systemd

`systemd` is a system and session manager for Linux, compatible with System V and LSB init scripts. The main features are:

- provides aggressive parallelization capabilities
- uses socket and D-Bus activation for starting services
- offers on-demand starting of daemons
- keeps track of processes using Linux cgroups
- supports snapshotting and restoring of the system state
- maintains mount and automount points
- implements an elaborate transactional dependency-based service control logic

13.1.2 Unit File

A unit configuration file encodes information about a service, a socket, a device, a mount point, an automount point, a swap file or partition, a start-up target, a watched file system path, a timer controlled and supervised by systemd, a temporary system state snapshot, a resource management slice or a group of externally created processes. “Unit file” is a generic term used by systemd for the following:

- **Service.** Information about a process (for example running a daemon); file ends with `.service`
- **Targets.** Used for grouping units and as synchronization points during start-up; file ends with `.target`
- **Sockets.** Information about an IPC or network socket or a file system FIFO, for socket-based activation (like `inetd`); file ends with `.socket`
- **Path.** Used to trigger other units (for example running a service when files change); file ends with `.path`
- **Timer.** Information about a timer controlled, for timer-based activation; file ends with `.timer`
- **Mount point.** Usually auto-generated by the `fstab` generator; file ends with `.mount`
- **Automount point.** Information about a file system automount point; file ends with `.automount`
- **Swap.** Information about a swap device or file for memory paging; file ends with `.swap`
- **Device.** Information about a device unit as exposed in the `sysfs/udev(7)` device tree; file ends with `.device`
- **Scope / Slice.** A concept for hierarchically managing resources of a group of processes; file ends with `.scope/.slice`

For more information about `systemd.unit` see <http://www.freedesktop.org/software/systemd/man/systemd.unit.html> ↗

13.2 Basic Usage

The System V init system uses several commands to handle services—the init scripts, insserv, telinit and others. systemd makes it easier to manage services, since there is only one command to memorize for the majority of service-handling tasks: systemctl. It uses the “command plus subcommand” notation like git or zypper:


```
systemctl [general OPTIONS] subcommand [subcommand OPTIONS]
```

See man 1 systemctl for a complete manual.



Tip: Terminal Output and Bash Completion

If the output goes to a terminal (and not to a pipe or a file, for example) systemd commands send long output to a pager by default. Use the --no-pager option to turn off paging mode.

systemd also supports bash-completion, allowing you to enter the first letters of a subcommand and then press  to automatically complete it. This feature is only available in the bash shell and requires the installation of the package bash-completion.

13.2.1 Managing Services in a Running System

Subcommands for managing services are the same as for managing a service with System V init (start, stop, ...). The general syntax for service management commands is as follows:

systemd

```
systemctl reload|restart|start|status|stop|... <my_service(s)>
```

System V init

```
rc<my_service(s)> reload|restart|start|status|stop|...
```

systemd allows you to manage several services in one go. Instead of executing init scripts one after the other as with System V init, execute a command like the following:

```
systemctl start <my_1st_service> <my_2nd_service>
```

If you want to list all services available on the system:

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

The following table lists the most important service management commands for systemd and System V init:

TABLE 13.1: SERVICE MANAGEMENT COMMANDS

Task	systemd Command	System V init Command
Starting.	start	start
Stopping.	stop	stop
Restarting. Shuts down services and starts them afterward. If a service is not yet running it will be started.	restart	restart
Restarting conditionally. Restarts services if they are currently running. Does nothing for services that are not running.	try-restart	try-restart
Reloading. Tells services to reload their configuration files without interrupting operation. Use case: Tell Apache to reload a modified <code>httpd.conf</code> configuration file. Note that not all services support reloading.	reload	reload
Reloading or restarting. Reloads services if reloading is supported, otherwise restarts them. If a service is not yet running it will be started.	reload-or-restart	n/a
Reloading or restarting conditionally. Reloads services if reloading is supported, otherwise restarts them if currently running. Does nothing for services that are not running.	reload-or-try-restart	n/a

Task	systemd Command	System V init Command
Getting detailed status information. Lists information about the status of services. The <code>systemd</code> command shows details such as description, executable, status, cgroup, and messages last issued by a service (see Section 13.6.8, “Debugging Services”). The level of details displayed with the System V init differs from service to service.	status	status
Getting short status information. Shows whether services are active or not.	is-active	status

13.2.2 Permanently Enabling/Disabling Services

The service management commands mentioned in the previous section let you manipulate services for the current session. `systemd` also lets you permanently enable or disable services, so they are automatically started when requested or are always unavailable. You can either do this by using YaST, or on the command line.

13.2.2.1 Enabling/Disabling Services on the Command Line

The following table lists enabling and disabling commands for `systemd` and System V init:

Important: Service Start

When enabling a service on the command line, it is not started automatically. It is scheduled to be started with the next system start-up or runlevel/target change. To immediately start a service after having enabled it, explicitly run `systemctl start <my_service>` or `rc <my_service> start`.

TABLE 13.2: COMMANDS FOR ENABLING AND DISABLING SERVICES

Task	<u>systemd</u> Command	System V init Command
Enabling.	<u>systemctl enable</u> <u><my_service(s)></u>	<u>insserv</u> <u><my_service(s)></u>
Disabling.	<u>systemctl disable</u> <u><my_service(s)>.service</u>	<u>insserv -r</u> <u><my_service(s)></u>
Checking. Shows whether a service is enabled or not.	<u>systemctl is-enabled</u> <u><my_service></u>	n/a
Re-enabling. Similar to restarting a service, this command first disables and then enables a service. Useful to re-enable a service with its defaults.	<u>systemctl reenab</u> <u><my_service></u>	n/a
Masking. After “disabling” a service, it can still be started manually. To completely disable a service, you need to mask it. Use with care.	<u>systemctl mask <my_service></u>	n/a
Unmasking. A service that has been masked can only be used again after it has been unmasked.	<u>systemctl unmask</u> <u><my_service></u>	n/a

13.3 System Start and Target Management

The entire process of starting the system and shutting it down is maintained by systemd. From this point of view, the Kernel can be considered a background process to maintain all other processes and adjust CPU time and hardware access according to requests from other programs.

13.3.1 Targets Compared to Runlevels

With System V init the system was booted into a so-called “Runlevel”. A runlevel defines how the system is started and what services are available in the running system. Runlevels are numbered; the most commonly known ones are 0 (shutting down the system), 3 (multiuser with network) and 5 (multiuser with network and display manager).

systemd introduces a new concept by using so-called “target units”. However, it remains fully compatible with the runlevel concept. Target units are named rather than numbered and serve specific purposes. For example, the targets local-fs.target and swap.target mount local file systems and swap spaces.

The target graphical.target provides a multiuser system with network and display manager capabilities and is equivalent to runlevel 5. Complex targets, such as graphical.target act as “meta” targets by combining a subset of other targets. Since systemd makes it easy to create custom targets by combining existing targets, it offers great flexibility.

The following list shows the most important systemd target units. For a full list refer to man 7 systemd.special.

SELECTED SYSTEMD TARGET UNITS

default.target

The target that is booted by default. Not a “real” target, but rather a symbolic link to another target like graphic.target. Can be permanently changed via YaST (see [Section 13.4, “Managing Services with YaST”](#)). To change it for a session, use the Kernel command line option systemd.unit=<my_target>.target at the boot prompt.

emergency.target

Starts an emergency shell on the console. Only use it at the boot prompt as systemd.unit=emergency.target.

graphical.target

Starts a system with network, multiuser support and a display manager.

halt.target

Shuts down the system.

mail-transfer-agent.target

Starts all services necessary for sending and receiving mails.

multi-user.target

Starts a multiuser system with network.

reboot.target

Reboots the system.

rescue.target

Starts a single-user system without network.

To remain compatible with the System V init runlevel system, systemd provides special targets named runlevelX.target mapping the corresponding runlevels numbered X.

If you want to know the current target, use the command: **systemctl get-default**

TABLE 13.3: SYSTEM V RUNLEVELS AND systemd TARGET UNITS

System V run-level	systemd target	Purpose
0	<u>runlevel0.target</u> , <u>halt.target</u> , <u>poweroff.target</u>	System shutdown
1, S	<u>runlevel1.target</u> , <u>rescue.target</u> ,	Single-user mode
2	<u>runlevel2.target</u> , <u>mul-</u> <u>ti-user.target</u> ,	Local multiuser without remote network
3	<u>runlevel3.target</u> , <u>mul-</u> <u>ti-user.target</u> ,	Full multiuser with network
4	<u>runlevel4.target</u>	Unused/User-defined
5	<u>runlevel5.target</u> , <u>graphical.target</u> ,	Full multiuser with network and display manager
6	<u>runlevel6.target</u> , <u>reboot.target</u> ,	System reboot



Important: systemd ignores /etc/inittab

The runlevels in a System V init system are configured in /etc/inittab. systemd does *not* use this configuration. Refer to [Section 13.5.3, “Creating Custom Targets”](#) for instructions on how to create your own bootable target.

13.3.1.1 Commands to Change Targets

Use the following commands to operate with target units:

Task	systemd Command	System V init Command
Change the current target/run-level	<u>systemctl isolate</u> <u><my_target></u> .target	<u>telinit</u> <u>X</u>
Change to the default target/runlevel	<u>systemctl default</u>	n/a
Get the current target/runlevel	<u>systemctl list-units --type=target</u> With systemd there is usually more than one active target. The command lists all currently active targets.	<u>who -r</u> or <u>runlevel</u>
persistently change the default runlevel	Use the Services Manager or run the following command: <u>ln -sf /usr/lib/systemd/system/</u> <u><my_target>.target /etc/systemd/system/default.target</u>	Use the Services Manager or change the line <u>id: X:initdefault:</u> <u>in /etc/inittab</u>
Change the default runlevel for the current boot process	Enter the following option at the boot prompt <u>systemd.unit=</u> <u><my_target></u> .target	Enter the desired run-level number at the boot prompt.
Show a target's/runlevel's dependencies	<u>systemctl show -p "Requires"</u> <u><my_target>.target</u> <u>systemctl show -p "Wants"</u> <u><my_target>.target</u> “Requires” lists the hard dependencies (the ones that must be resolved), whereas “Wants” lists the soft dependencies (the ones that get resolved if possible).	n/a

13.3.2 Debugging System Start-Up

systemd offers the means to analyze the system start-up process. You can conveniently review the list of all services and their status (rather than having to parse `/varlog/`). systemd also allows you to scan the start-up procedure to find out how much time each service start-up consumes.

13.3.2.1 Review Start-Up of Services

To review the complete list of services that have been started since booting the system, enter the command **systemctl**. It lists all active services like shown below (shortened). To get more information on a specific service, use **systemctl status** *<my_service>*.

EXAMPLE 13.1: LIST ACTIVE SERVICES

```
root # systemctl
UNIT                                LOAD    ACTIVE SUB    JOB DESCRIPTION
[...]
iscsi.service                      loaded active exited  Login and scanning of iSC+
kmod-static-nodes.service          loaded active exited  Create list of required s+
libvirtd.service                   loaded active running  Virtualization daemon
nscd.service                       loaded active running  Name Service Cache Daemon
ntpd.service                       loaded active running  NTP Server Daemon
polkit.service                     loaded active running  Authorization Manager
postfix.service                    loaded active running  Postfix Mail Transport Ag+
rc-local.service                   loaded active exited  /etc/init.d/boot.local Co+
rsyslog.service                    loaded active running  System Logging Service
[...]
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.

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

To restrict the output to services that failed to start, use the **--failed** option:

EXAMPLE 13.2: LIST FAILED SERVICES

```
root # systemctl --failed
UNIT                                LOAD    ACTIVE SUB    JOB DESCRIPTION
apache2.service                    loaded failed failed    apache
NetworkManager.service             loaded failed failed    Network Manager
```

```
plymouth-start.service loaded failed failed      Show Plymouth Boot Screen

[...]
```

13.3.2.2 Debug Start-Up Time

To debug system start-up time, systemd offers the **systemd-analyze** command. It shows the total start-up time, a list of services ordered by start-up time and can also generate an SVG graphic showing the time services took to start in relation to the other services.

Listing the System Start-Up Time

```
root # systemd-analyze
Startup finished in 2666ms (kernel) + 21961ms (userspace) = 24628ms
```

Listing the Services Start-Up Time

```
root # systemd-analyze blame
6472ms systemd-modules-load.service
5833ms remount-rootfs.service
4597ms network.service
4254ms systemd-vconsole-setup.service
4096ms postfix.service
2998ms xdm.service
2483ms localnet.service
2470ms SuSEfirewall2_init.service
2189ms avahi-daemon.service
2120ms systemd-logind.service
1210ms xinetd.service
1080ms ntp.service
[...]
75ms fbset.service
72ms purge-kernels.service
47ms dev-vdal.swap
38ms bluez-coldplug.service
35ms splash_early.service
```

Services Start-Up Time Graphics

```
root # systemd-analyze plot > jupiter.example.com-startup.svg
```


13.3.3 System V Compatibility

systemd is compatible with System V, allowing you to still use existing System V init scripts. However, there is at least one known issue where a System V init script does not work with systemd out of the box: starting a service as a different user via su or sudo in init scripts will result in a failure of the script, producing an “Access denied” error.

When changing the user with su or sudo, a PAM session is started. This session will be terminated after the init script is finished. As a consequence, the service that has been started by the init script will also be terminated. To work around this error, proceed as follows:

1. Create a service file wrapper with the same name as the init script plus the file name extension .service:

```
[Unit]
Description=DESCRIPTION
After=network.target

[Service]
User=USER
Type=forking ❶
PIDFile=PATH TO PID FILE ❶
ExecStart=PATH TO INIT SCRIPT start
ExecStop=PATH TO INIT SCRIPT stop
ExecStopPost=/usr/bin/rm -f PATH TO PID FILE ❶

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

Replace all values written in UPPERCASE LETTERS with appropriate values.

- ❶ Optional—only use if the init script starts a daemon.
- ❷ multi-user.target also starts the init script when booting into graphical.target. If it should only be started when booting into the display manager, use graphical.target here.

2. Start the daemon with **systemctl start APPLICATION**.

13.4 Managing Services with YaST

Basic service management can also be done with the YaST Services Manager module. It supports starting, stopping, enabling and disabling services. It also lets you show a service's status and change the default target. Start the YaST module with *YaST > System > Services Manager*.

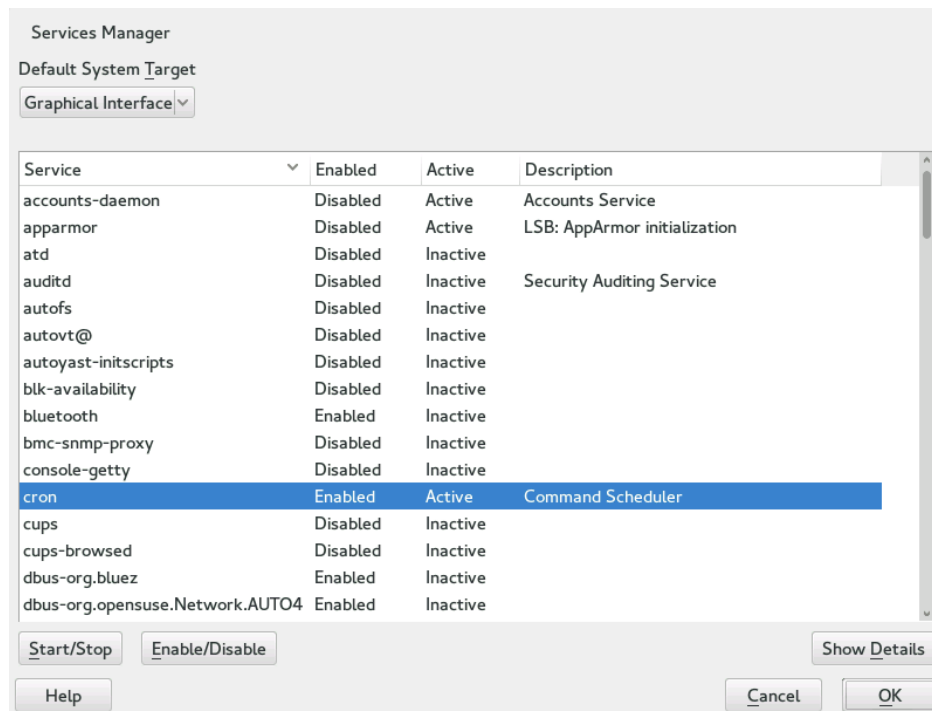


FIGURE 13.1: SERVICES MANAGER

Changing the *Default System Target*

To change the target the system boots into, choose a target from the *Default System Target* drop-down box. The most often used targets are *Graphical Interface* (starting a graphical login screen) and *Multi-User* (starting the system in command line mode).

Starting or Stopping a Service

Select a service from the table. The *Active* column shows whether it is currently running (*Active*) or not (*Inactive*). Toggle its status by choosing *Start/Stop*.

Starting or stopping a service changes its status for the currently running session. To change its status throughout a reboot, you need to enable or disable it.

Enabling or Disabling a Service

Select a service from the table. The *Enabled* column shows whether it is currently *Enabled* or *Disabled*. Toggle its status by choosing *Enable/Disable*.

By enabling or disabling a service you configure whether it is started during booting (*Enabled*) or not (*Disabled*). This setting will not affect the current session. To change its status in the current session, you need to start or stop it.

View a Status Messages

To view the status message of a service, select it from the list and choose *Show Details*. The output you will see is identical to the one generated by the command `systemctl -l status <my_service>`.



Warning: Faulty Runlevel Settings May Damage Your System

Faulty runlevel settings may make your system unusable. Before applying your changes, make absolutely sure that you know their consequences.

13.5 Customization of systemd

The following sections contain some examples for `systemd` customization.



Warning: Avoiding Overwritten Customization

Always do `systemd` customization in `/etc/systemd/`, *never* in `/usr/lib/systemd/`. Otherwise your changes will be overwritten by the next update of `systemd`.

13.5.1 Customizing Service Files

The `systemd` service files are located in `/usr/lib/systemd/system`. If you want to customize them, proceed as follows:

1. Copy the files you want to modify from `/usr/lib/systemd/system` to `/etc/systemd/system`. Keep the file names identical to the original ones.
2. Modify the copies in `/etc/systemd/system` according to your needs.
3. For an overview of your configuration changes, use the `systemd-delta` command. It can compare and identify configuration files that override other configuration files. For details, refer to the `systemd-delta` man page.

The modified files in /etc/systemd will take precedence over the original files in /usr/lib/systemd/system, provided that their file name is the same.

13.5.2 Creating “Drop-in” Files

If you only want to add a few lines to a configuration file or modify a small part of it, you can use so-called “drop-in” files. Drop-in files let you extend the configuration of unit files without having to edit or override the unit files themselves.

For example, to change one value for the foobar service located in /usr/lib/systemd/system/foobar.service, proceed as follows:

1. Create a directory called /etc/systemd/system/<my_service>.service.d/.
Note the .d suffix. The directory must otherwise be named like the service that you want to patch with the drop-in file.
2. In that directory, create a file whatevermodification.conf.
Make sure it only contains the line with the value that you want to modify.
3. Save your changes to the file. It will be used as an extension of the original file.

13.5.3 Creating Custom Targets

On System V init SUSE systems, runlevel 4 is unused to allow administrators to create their own runlevel configuration. systemd allows you to create any number of custom targets. It is suggested to start by adapting an existing target such as graphical.target.

1. Copy the configuration file /usr/lib/systemd/system/graphical.target to /etc/systemd/system/<my_target>.target and adjust it according to your needs.
2. The configuration file copied in the previous step already covers the required (“hard”) dependencies for the target. To also cover the wanted (“soft”) dependencies, create a directory /etc/systemd/system/<my_target>.target.wants.
3. For each wanted service, create a symbolic link from /usr/lib/systemd/system into /etc/systemd/system/<my_target>.target.wants.

4. Once you have finished setting up the target, reload the systemd configuration to make the new target available:

```
systemctl daemon-reload
```

13.6 Advanced Usage

The following sections cover advanced topics for system administrators. For even more advanced systemd documentation, refer to Lennart Pöttering's series about systemd for administrators at <http://0pointer.de/blog/projects> ↗.

13.6.1 Cleaning Temporary Directories

`systemd` supports cleaning temporary directories regularly. The configuration from the previous system version is automatically migrated and active. `tmpfiles.d`—which is responsible for managing temporary files—reads its configuration from `/etc/tmpfiles.d/*.conf`, `/run/tmpfiles.d/*.conf`, and `/usr/lib/tmpfiles.d/*.conf` files. Configuration placed in `/etc/tmpfiles.d/*.conf` overrides related configurations from the other two directories (`/usr/lib/tmpfiles.d/*.conf` is where packages store their configuration files).

The configuration format is one line per path containing action and path, and optionally mode, ownership, age and argument fields, depending on the action. The following example unlinks the X11 lock files:

Type	Path	Mode	UID	GID	Age	Argument
r	/tmp/.X[0-9]*-lock					

To get the status the tmpfile timer:

```
systemctl status systemd-tmpfiles-clean.timer
systemd-tmpfiles-clean.timer - Daily Cleanup of Temporary Directories
Loaded: loaded (/usr/lib/systemd/system/systemd-tmpfiles-clean.timer; static)
Active: active (waiting) since Tue 2014-09-09 15:30:36 CEST; 1 weeks 6 days ago
Docs: man:tmpfiles.d(5)
      man:systemd-tmpfiles(8)

Sep 09 15:30:36 jupiter systemd[1]: Starting Daily Cleanup of Temporary Directories.
Sep 09 15:30:36 jupiter systemd[1]: Started Daily Cleanup of Temporary Directories.
```

For more information on temporary files handling, see [man 5 tmpfiles.d](#).

13.6.2 System Log

Section 13.6.8, “Debugging Services” explains how to view log messages for a given service. However, displaying log messages is not restricted to service logs. You can also access and query the complete log messages written by `systemd`—the so-called “Journal”. Use the command `systemd-journalctl` to display the complete log messages starting with the oldest entries. Refer to [man 1 systemd-journalctl](#) for options such as applying filters or changing the output format.

13.6.3 Snapshots

You can save the current state of `systemd` to a named snapshot and later revert to it with the `isolate` subcommand. This is useful when testing services or custom targets, because it allows you to return to a defined state at any time. A snapshot is only available in the current session and will automatically be deleted on reboot. A snapshot name must end in `.snapshot`.

Create a Snapshot

```
systemctl snapshot <my_snapshot>.snapshot
```

Delete a Snapshot

```
systemctl delete <my_snapshot>.snapshot
```

View a Snapshot

```
systemctl show <my_snapshot>.snapshot
```

Activate a Snapshot

```
systemctl isolate <my_snapshot>.snapshot
```

13.6.4 Loading Kernel Modules

With `systemd`, kernel modules can automatically be loaded at boot time via a configuration file in `/etc/modules-load.d`. The file should be named `module.conf` and have the following content:

```
# load module module at boot time
module
```

In case a package installs a configuration file for loading a Kernel module, the file gets installed to `/usr/lib/modules-load.d`. If two configuration files with the same name exist, the one in `/etc/modules-load.d` takes precedence.

For more information, see the `modules-load.d(5)` man page.

13.6.5 Performing Actions Before Loading a Service

With System V init actions that need to be performed before loading a service, needed to be specified in `/etc/init.d/before.local`. This procedure is no longer supported with `systemd`. If you need to do actions before starting services, do the following:

Loading Kernel Modules

Create a drop-in file in `/etc/modules-load.d` directory (see `man modules-load.d` for the syntax)

Creating Files or Directories, Cleaning-up Directories, Changing Ownership

Create a drop-in file in `/etc/tmpfiles.d` (see `man tmpfiles.d` for the syntax)

Other Tasks

Create a system service file, for example `/etc/systemd/system/before.service`, from the following template:

```
[Unit]
Before=NAME OF THE SERVICE YOU WANT THIS SERVICE TO BE STARTED BEFORE
[Service]
Type=oneshot
RemainAfterExit=true
ExecStart=YOUR_COMMAND
# beware, executable is run directly, not through a shell, check the man pages
# systemd.service and systemd.unit for full syntax
[Install]
# target in which to start the service
```

```
WantedBy=multi-user.target
#WantedBy=graphical.target
```

When the service file is created, you should run the following commands (as root):

```
systemctl daemon-reload
systemctl enable before
```

Every time you modify the service file, you need to run:

```
systemctl daemon-reload
```

13.6.6 Kernel Control Groups (cgroups)

On a traditional System V init system it is not always possible to clearly assign a process to the service that spawned it. Some services, such as Apache, spawn a lot of third-party processes (for example CGI or Java processes), which themselves spawn more processes. This makes a clear assignment difficult or even impossible. Additionally, a service may not terminate correctly, leaving some children alive.

systemd solves this problem by placing each service into its own cgroup. cgroups are a Kernel feature that allows aggregating processes and all their children into hierarchical organized groups. systemd names each cgroup after its service. Since a non-privileged process is not allowed to “leave” its cgroup, this provides an effective way to label all processes spawned by a service with the name of the service.

To list all processes belonging to a service, use the command **systemd-cgls**. The result will look like the following (shortened) example:

EXAMPLE 13.3: LIST ALL PROCESSES BELONGING TO A SERVICE

```
root # systemd-cgls --no-pager
├─1 /usr/lib/systemd/systemd --switched-root --system --deserialize 20
├─user.slice
│   └─user-1000.slice
│       └─session-102.scope
│           ├──12426 gdm-session-worker [pam/gdm-password]
│           ├──15831 gdm-session-worker [pam/gdm-password]
│           ├──15839 gdm-session-worker [pam/gdm-password]
│           └─15858 /usr/lib/gnome-terminal-server
[...]
```



```

└─system.slice
  ├─systemd-hostnamed.service
  │   └─17616 /usr/lib/systemd/systemd-hostnamed
  ├─cron.service
  │   └─1689 /usr/sbin/cron -n
  ├─ntpd.service
  │   └─1328 /usr/sbin/ntpd -p /var/run/ntp/ntpd.pid -g -u ntp:ntp -c /etc/ntp.conf
  ├─postfix.service
  │   ├─1676 /usr/lib/postfix/master -w
  │   ├─1679 qmgr -l -t fifo -u
  │   └─15590 pickup -l -t fifo -u
  ├─sshd.service
  │   └─1436 /usr/sbin/sshd -D
  [...]

```

See Book “System Analysis and Tuning Guide”, Chapter 9 “Kernel Control Groups” for more information about cgroups.

13.6.7 Terminating Services (Sending Signals)

As explained in [Section 13.6.6, “Kernel Control Groups \(cgroups\)”](#), it is not always possible to assign a process to its parent service process in a System V init system. This makes it difficult to terminate a service and all of its children. Child processes that have not been terminated will remain as zombie processes.

systemd's concept of confining each service into a cgroup makes it possible to clearly identify all child processes of a service and therefore allows you to send a signal to each of these processes. Use **systemctl kill** to send signals to services. For a list of available signals refer to [man 7 signals](#).

Sending SIGTERM to a Service

SIGTERM is the default signal that is sent.

```
systemctl kill <my_service>
```

Sending SIGNAL to a Service

Use the -s option to specify the signal that should be sent.

```
systemctl kill -s SIGNAL <my_service>
```

Selecting Processes

By default the **kill** command sends the signal to all processes of the specified cgroup. You can restrict it to the control or the main process. The latter is for example useful to force a service to reload its configuration by sending SIGHUP:

```
systemctl kill -s SIGHUP --kill-who=main <my_service>
```

13.6.8 Debugging Services

By default, systemd is not overly verbose. If a service was started successfully, no output will be produced. In case of a failure, a short error message will be displayed. However, **systemctl status** provides means to debug start-up and operation of a service.

systemd comes with its own logging mechanism (“The Journal”) that logs system messages. This allows you to display the service messages together with status messages. The **status** command works similar to **tail** and can also display the log messages in different formats, making it a powerful debugging tool.

Show Service Start-Up Failure

Whenever a service fails to start, use **systemctl status <my_service>** to get a detailed error message:

```
root # systemctl start apache2
Job failed. See system journal and 'systemctl status' for details.
root # systemctl status apache2
    Loaded: loaded (/usr/lib/systemd/system/apache2.service; disabled)
    Active: failed (Result: exit-code) since Mon, 04 Jun 2012 16:52:26 +0200;
    29s ago
    Process: 3088 ExecStart=/usr/sbin/start_apache2 -D SYSTEMD -k start
    (code=exited, status=1/FAILURE)
    CGroup: name=systemd:/system/apache2.service

Jun 04 16:52:26 g144 start_apache2[3088]: httpd2-prefork: Syntax error on line
205 of /etc/apache2/httpd.conf: Syntax error on li...alHost>
```

Show Last *n* Service Messages

The default behavior of the **status** subcommand is to display the last ten messages a service issued. To change the number of messages to show, use the **--lines=*n*** parameter:

```
systemctl status ntp
```

```
systemctl --lines=20 status ntp
```

Show Service Messages in Append Mode

To display a “live stream” of service messages, use the `--follow` option, which works like `tail -f`:

```
systemctl --follow status ntp
```

Messages Output Format

The `--output=mode` parameter allows you to change the output format of service messages. The most important modes available are:

short

The default format. Shows the log messages with a human readable time stamp.

verbose

Full output with all fields.

cat

Terse output without time stamps.

13.7 More Information

For more information on systemd refer to the following online resources:

Homepage

<http://www.freedesktop.org/wiki/Software/systemd> ↗

systemd for Administrators

Lennart Pöttering, one of the systemd authors, has written a series of blog entries (13 at the time of writing this chapter). Find them at <http://0pointer.de/blog/projects> ↗.

14 journalctl: Query the systemd Journal

When `systemd` replaced traditional init scripts in SUSE Linux Enterprise 12 (see [Chapter 13, The systemd Daemon](#)), it introduced its own logging system called *journal*. There is no need to run a `syslog` based service anymore, as all system events are written in the journal.

The journal itself is a system service managed by `systemd`. Its full name is `systemd-journald.service`. It collects and stores logging data by maintaining structured indexed journals based on logging information received from the kernel, from user processes, from standard input and from system service errors. The `systemd-journald` service is on by default:

```
# systemctl status systemd-journald
systemd-journald.service - Journal Service
   Loaded: loaded (/usr/lib/systemd/system/systemd-journald.service; static)
   Active: active (running) since Mon 2014-05-26 08:36:59 EDT; 3 days ago
     Docs: man:systemd-journald.service(8)
           man:journald.conf(5)
  Main PID: 413 (systemd-journal)
    Status: "Processing requests..."
   CGroup: /system.slice/systemd-journald.service
           └─413 /usr/lib/systemd/systemd-journald
[...]
```

14.1 Making the Journal Persistent

The journal stores log data in `/run/log/journal/` by default. Because the `/run/` directory is volatile by nature, log data is lost at reboot. To make the log data persistent, the directory `/var/log/journal/` with correct ownership and permissions must exist, where the `systemd-journald` service can store its data. `systemd` will create the directory for you—and switch to persistent logging—if you do the following:

1. As `root`, open `/etc/systemd/journald.conf` for editing.

```
# vi /etc/systemd/journald.conf
```

2. Uncomment the line containing `Storage=` and change it to

```
[...]
[Journal]
```

```
Storage=persistent
#Compress=yes
[...]
```

3. Save the file and restart systemd-journald:

```
systemctl restart systemd-journald
```

14.2 journalctl Useful Switches

This section introduces several common useful options to enhance the default **journalctl** behavior. All switches are described in the **journalctl** manual page, **man 1 journalctl**.



Tip: Messages Related to a Specific Executable

To show all journal messages related to a specific executable, specify the full path to the executable:

```
journalctl /usr/lib/systemd/systemd
```

-f

Shows only the most recent journal messages, and prints new log entries as they are added to the journal.

-e

Prints the messages and jumps to the end of the journal, so that the latest entries are visible within the pager.

-r

Prints the messages of the journal in reverse order, so that the latest entries are listed first.

-k

Shows only kernel messages. This is equivalent to the field match **__TRANSPORT=kernel** (see [Section 14.3.3, "Filtering Based on Fields"](#)).

-u

Shows only messages for the specified **systemd** unit. This is equivalent to the field match **__SYSTEMD_UNIT=UNIT** (see [Section 14.3.3, "Filtering Based on Fields"](#)).

```
# journalctl -u apache2
[...]
Jun 03 10:07:11 pinkiepie systemd[1]: Starting The Apache Webserver...
Jun 03 10:07:12 pinkiepie systemd[1]: Started The Apache Webserver.
```

14.3 Filtering the Journal Output

When called without switches, **journalctl** shows the full content of the journal, the oldest entries listed first. The output can be filtered by specific switches and fields.

14.3.1 Filtering Based on a Boot Number

journalctl can filter messages based on a specific system boot. To list all available boots, run

```
# journalctl --list-boots
-1 097ed2cd99124a2391d2cfffab1b566f0 Mon 2014-05-26 08:36:56 EDT-Fri 2014-05-30
05:33:44 EDT
0 156019a44a774a0bb0148a92df4af81b Fri 2014-05-30 05:34:09 EDT-Fri 2014-05-30
06:15:01 EDT
```

The first column lists the boot offset: 0 for the current boot, -1 for the previous, -2 for the prior to that, etc. The second column contains the boot ID, and then the limiting time stamps of the specific boot follow.

Show all messages from the current boot:

```
# journalctl -b
```

If you need to see journal messages from the previous boot, add an offset parameter. The following example outputs the previous boot messages:

```
# journalctl -b -1
```

Another way is to list boot messages based on the boot ID. For this purpose, use the `_BOOT_ID` field:

```
# journalctl _BOOT_ID=156019a44a774a0bb0148a92df4af81b
```

14.3.2 Filtering Based on Time Interval

You can filter the output of **journalctl** by specifying the starting and/or ending date. The date specification should be of the format "2014-06-30 9:17:16". If the time part is omitted, midnight is assumed. If seconds are omitted, ":00" is assumed. If the date part is omitted, the current day is assumed. Instead of numeric expression, you can specify the keywords "yesterday", "today", or "tomorrow", which refer to midnight of the day before the current day, of the current day, or of the day after the current day. If you specify "now", it refers to the current time. You can also specify relative times prefixed with **-** or **+**, referring to times before or after the current time. Show only new messages since now, and update the output continuously:

```
# journalctl --since "now" -f
```

Show all messages since last midnight till 3:20am:

```
# journalctl --since "today" --until "3:20"
```

14.3.3 Filtering Based on Fields

You can filter the output of the journal by specific fields. The syntax of a field to be matched is **FIELD_NAME=MATCHED_VALUE**, such as **_SYSTEMD_UNIT=httpd.service**. You can specify multiple matches in a single query to filter the output messages even more. See **man 7 systemd.journal-fields** for a list of default fields.

Show messages produced by a specific process ID:

```
# journalctl _PID=1039
```

Show messages belonging to a specific user ID:

```
# journalctl _UID=1000
```

Show messages from the kernel ring buffer (the same as **dmesg** produces):

```
# journalctl _TRANSPORT=kernel
```

Show messages from the service's standard or error output:

```
# journalctl _TRANSPORT=stdout
```

Show messages produced by a specified service only:

```
# journalctl _SYSTEMD_UNIT=avahi-daemon.service
```

If two different fields are specified, only entries that match both expressions at the same time are shown:

```
# journalctl _SYSTEMD_UNIT=avahi-daemon.service _PID=1488
```

If two matches refer to the same field, all entries matching either expression are shown:

```
# journalctl _SYSTEMD_UNIT=avahi-daemon.service _SYSTEMD_UNIT=dbus.service
```

You can use the '+' separator to combine two expressions in a logical 'OR'. The following example shows all messages from the Avahi service process with the process ID 1480 together with all messages from the D-Bus service:

```
# journalctl _SYSTEMD_UNIT=avahi-daemon.service _PID=1480 +  
_SYSTEMD_UNIT=dbus.service
```

14.4 Investigating systemd Errors

This section introduces a simple example to illustrate how to find and fix the error reported by systemd during apache2 start-up.

1. Try to start the apache2 service:

```
# systemctl start apache2  
Job for apache2.service failed. See 'systemctl status apache2' and 'journalctl  
-xn' for details.
```

2. Let us see what the service's status says:

```
# systemctl status apache2  
apache2.service - The Apache Webserver  
   Loaded: loaded (/usr/lib/systemd/system/apache2.service; disabled)  
   Active: failed (Result: exit-code) since Tue 2014-06-03 11:08:13 CEST; 7min  
   ago  
  Process: 11026 ExecStop=/usr/sbin/start_apache2 -D SYSTEMD -DFOREGROUND \  
           -k graceful-stop (code=exited, status=1/FAILURE)
```


The ID of the process causing the failure is 11026.

3. Show the verbose version of messages related to process ID 11026:

```
# journalctl -o verbose _PID=11026
[...]
MESSAGE=AH00526: Syntax error on line 6 of /etc/apache2/default-server.conf:
[...]
MESSAGE=Invalid command 'DocumenttRoot', perhaps misspelled or defined by a
module
[...]
```

4. Fix the typo inside `/etc/apache2/default-server.conf`, start the apache2 service, and print its status:

```
# systemctl start apache2 && systemctl status apache2
apache2.service - The Apache Webserver
   Loaded: loaded (/usr/lib/systemd/system/apache2.service; disabled)
   Active: active (running) since Tue 2014-06-03 11:26:24 CEST; 4ms ago
   Process: 11026 ExecStop=/usr/sbin/start_apache2 -D SYSTEMD -DFOREGROUND
            -k graceful-stop (code=exited, status=1/FAILURE)
   Main PID: 11263 (httpd2-prefork)
   Status: "Processing requests..."
   CGroup: /system.slice/apache2.service
           └─11263 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D [...]
           └─11280 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D [...]
           └─11281 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D [...]
           └─11282 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D [...]
           └─11283 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D [...]
           └─11285 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D [...]
```

14.5 Journald Configuration

The behavior of the `systemd-journald` service can be adjusted by modifying `/etc/systemd/journald.conf`. This section introduces only basic option settings. For a complete file description, see [**man 5 journald.conf**](#). Note that you need to restart the journal for the changes to take effect with

```
# systemctl restart systemd-journald
```

14.5.1 Changing the Journal Size Limit

If the journal log data is saved to a persistent location (see [Section 14.1, “Making the Journal Persistent”](#)), it uses up to 10% of the file system the `/var/log/journal` resides on. For example, if `/var/log/journal` is located on a 30 GB `/var` partition, the journal may use up to 3 GB of the disk space. To change this limit, change (and uncomment) the `SystemMaxUse` option:

```
SystemMaxUse=50M
```

14.5.2 Forwarding the Journal to `/dev/ttyX`

You can forward the journal to a terminal device to inform you about system messages on a preferred terminal screen, for example `/dev/tty12`. Change the following `journal` options to

```
ForwardToConsole=yes
TTYPath=/dev/tty12
```

14.5.3 Forwarding the Journal to Syslog Facility

Journald is backward compatible with traditional syslog implementations such as `rsyslog`. Make sure the following is valid:

- `rsyslog` is installed.

```
# rpm -q rsyslog
rsyslog-7.4.8-2.16.x86_64
```

- `rsyslog` service is enabled.

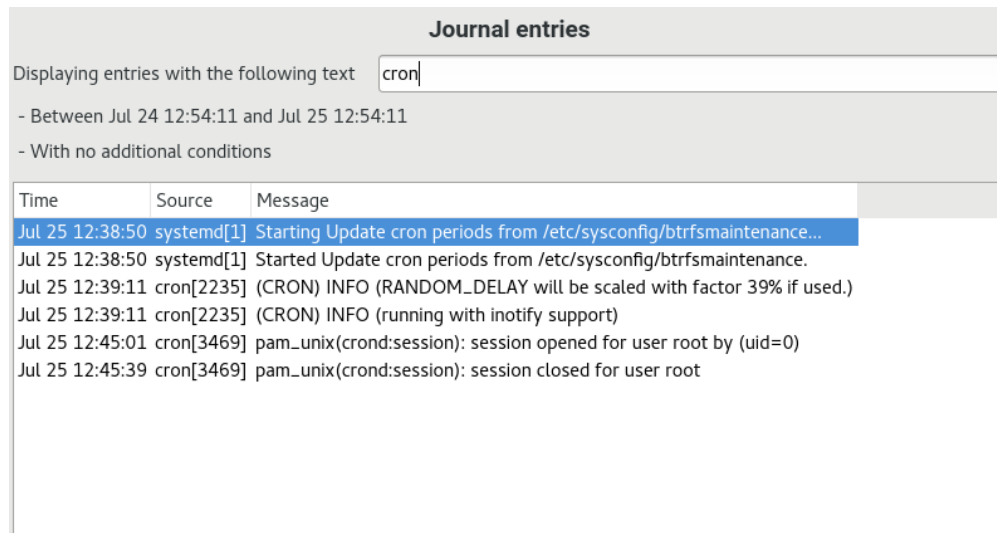
```
# systemctl is-enabled rsyslog
enabled
```

- Forwarding to syslog is enabled in `/etc/systemd/journal.conf`.

```
ForwardToSyslog=yes
```

14.6 Using YaST to Filter the systemd Journal

For an easy way of filtering the systemd journal (without having to deal with the `journalctl` syntax), you can use the YaST journal module. After installing it with **`sudo zypper in yast2-journal`**, start it from YaST by selecting *System > Systemd Journal*. Alternatively, start it from command line by entering **`sudo yast2 journal`**.



Displaying entries with the following text <input type="text" value="cron"/>		
- Between Jul 24 12:54:11 and Jul 25 12:54:11		
- With no additional conditions		
Time	Source	Message
Jul 25 12:38:50	systemd[1]	Starting Update cron periods from /etc/sysconfig/btrfsmaintenance...
Jul 25 12:38:50	systemd[1]	Started Update cron periods from /etc/sysconfig/btrfsmaintenance.
Jul 25 12:39:11	cron[2235]	(CRON) INFO (RANDOM_DELAY will be scaled with factor 39% if used.)
Jul 25 12:39:11	cron[2235]	(CRON) INFO (running with inotify support)
Jul 25 12:45:01	cron[3469]	pam_unix(cron:session): session opened for user root by (uid=0)
Jul 25 12:45:39	cron[3469]	pam_unix(cron:session): session closed for user root


FIGURE 14.1: YAST SYSTEMD JOURNAL

The module displays the log entries in a table. The search box on top allows you to search for entries that contain certain characters, similar to using **`grep`**. To filter the entries by date and time, unit, file, or priority, click *Change filters* and set the respective options.

15 Basic Networking

Linux offers the necessary networking tools and features for integration into all types of network structures. Network access using a network card can be configured with YaST. Manual configuration is also possible. In this chapter only the fundamental mechanisms and the relevant network configuration files are covered.

Linux and other Unix operating systems use the TCP/IP protocol. It is not a single network protocol, but a family of network protocols that offer various services. The protocols listed in *Several Protocols in the TCP/IP Protocol Family*, are provided for exchanging data between two machines via TCP/IP. Networks combined by TCP/IP, comprising a worldwide network, are also called “the Internet.”

RFC stands for *Request for Comments*. RFCs are documents that describe various Internet protocols and implementation procedures for the operating system and its applications. The RFC documents describe the setup of Internet protocols. For more information about RFCs, see <http://www.ietf.org/rfc.html> .

SEVERAL PROTOCOLS IN THE TCP/IP PROTOCOL FAMILY

TCP

Transmission Control Protocol: a connection-oriented secure protocol. The data to transmit is first sent by the application as a stream of data and converted into the appropriate format by the operating system. The data arrives at the respective application on the destination host in the original data stream format it was initially sent. TCP determines whether any data has been lost or jumbled during the transmission. TCP is implemented wherever the data sequence matters.

UDP

User Datagram Protocol: a connectionless, insecure protocol. The data to transmit is sent in the form of packets generated by the application. The order in which the data arrives at the recipient is not guaranteed and data loss is possible. UDP is suitable for record-oriented applications. It features a smaller latency period than TCP.

ICMP

Internet Control Message Protocol: Essentially, this is not a protocol for the end user, but a special control protocol that issues error reports and can control the behavior of machines participating in TCP/IP data transfer. In addition, it provides a special echo mode that can be viewed using the program ping.

IGMP

Internet Group Management Protocol: This protocol controls machine behavior when implementing IP multicast.

As shown in *Figure 15.1, "Simplified Layer Model for TCP/IP"*, data exchange takes place in different layers. The actual network layer is the insecure data transfer via IP (Internet protocol). On top of IP, TCP (transmission control protocol) guarantees, to a certain extent, security of the data transfer. The IP layer is supported by the underlying hardware-dependent protocol, such as Ethernet.

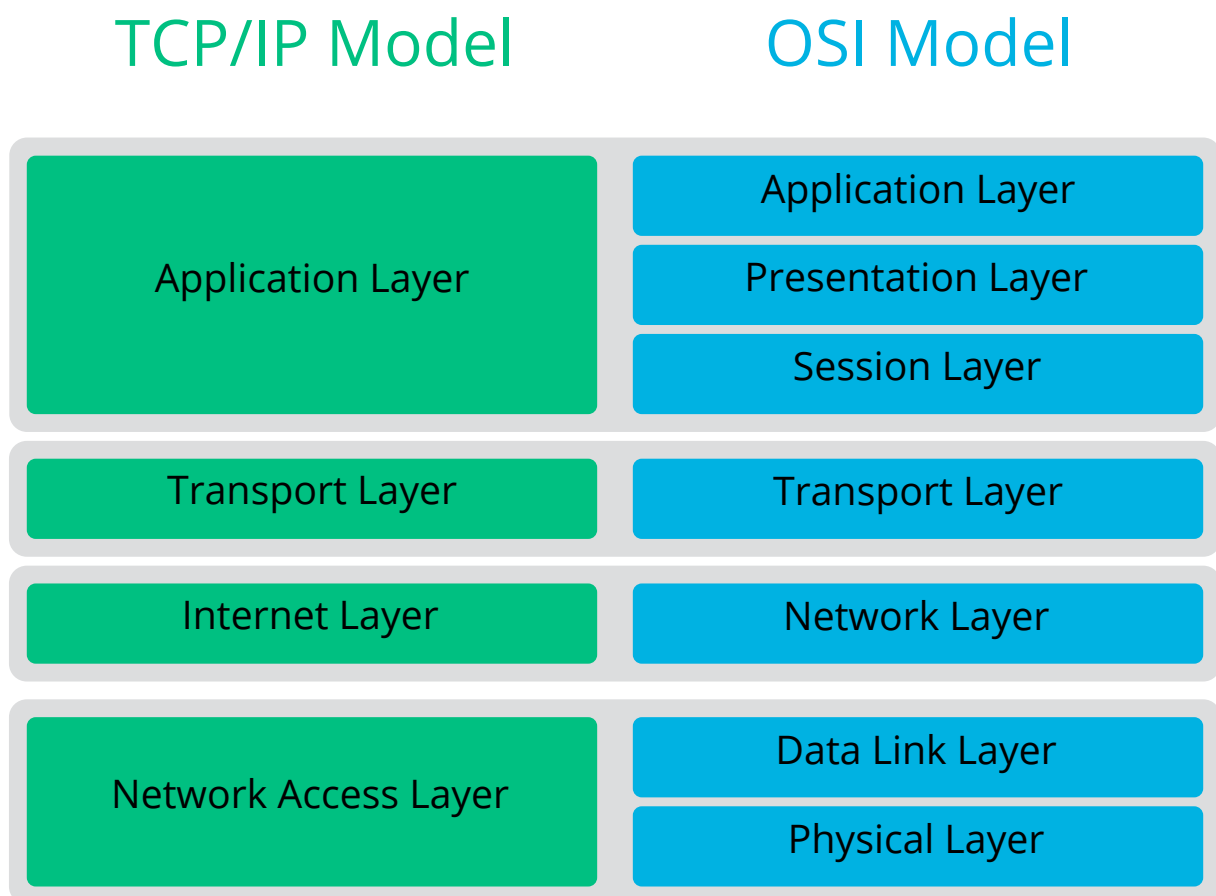


FIGURE 15.1: SIMPLIFIED LAYER MODEL FOR TCP/IP

The diagram provides one or two examples for each layer. The layers are ordered according to *abstraction levels*. The lowest layer is very close to the hardware. The uppermost layer, however, is almost a complete abstraction from the hardware. Every layer has its own special function. The special functions of each layer are mostly implicit in their description. The data link and physical layers represent the physical network used, such as Ethernet.

Almost all hardware protocols work on a packet-oriented basis. The data to transmit is collected into *packets* (it cannot be sent all at once). The maximum size of a TCP/IP packet is approximately 64 KB. Packets are normally quite smaller, as the network hardware can be a limiting factor. The maximum size of a data packet on an Ethernet is about fifteen hundred bytes. The size of a TCP/IP packet is limited to this amount when the data is sent over an Ethernet. If more data is transferred, more data packets need to be sent by the operating system.

For the layers to serve their designated functions, additional information regarding each layer must be saved in the data packet. This takes place in the *header* of the packet. Every layer attaches a small block of data, called the protocol header, to the front of each emerging packet. A sample TCP/IP data packet traveling over an Ethernet cable is illustrated in [Figure 15.2, “TCP/IP Ethernet Packet”](#). The proof sum is located at the end of the packet, not at the beginning. This simplifies things for the network hardware.

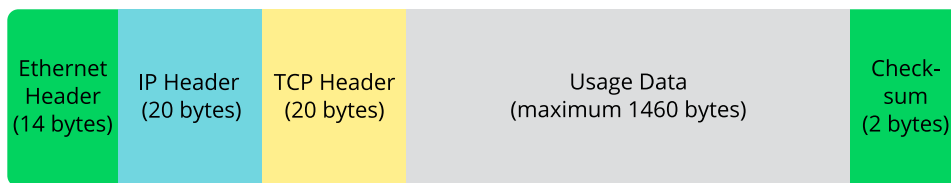


FIGURE 15.2: TCP/IP ETHERNET PACKET

When an application sends data over the network, the data passes through each layer, all implemented in the Linux Kernel except the physical layer. Each layer is responsible for preparing the data so it can be passed to the next layer. The lowest layer is ultimately responsible for sending the data. The entire procedure is reversed when data is received. Like the layers of an onion, in each layer the protocol headers are removed from the transported data. Finally, the transport layer is responsible for making the data available for use by the applications at the destination. In this manner, one layer only communicates with the layer directly above or below it. For applications, it is irrelevant whether data is transmitted via a 100 Mbit/s FDDI network or via a 56-Kbit/s modem line. Likewise, it is irrelevant for the data line which kind of data is transmitted, as long as packets are in the correct format.

15.1 IP Addresses and Routing

The discussion in this section is limited to IPv4 networks. For information about IPv6 protocol, the successor to IPv4, refer to [Section 15.2, “IPv6—The Next Generation Internet”](#).

15.1.1 IP Addresses

Every computer on the Internet has a unique 32-bit address. These 32 bits (or 4 bytes) are normally written as illustrated in the second row in [Example 15.1, “Writing IP Addresses”](#).

EXAMPLE 15.1: WRITING IP ADDRESSES

IP Address (binary):	11000000	10101000	00000000	00010100
IP Address (decimal):	192.	168.	0.	20

In decimal form, the four bytes are written in the decimal number system, separated by periods. The IP address is assigned to a host or a network interface. It can be used only once throughout the world. There are exceptions to this rule, but these are not relevant to the following passages. The points in IP addresses indicate the hierarchical system. Until the 1990s, IP addresses were strictly categorized in classes. However, this system proved too inflexible and was discontinued. Now, *classless routing* (CIDR, classless interdomain routing) is used.

15.1.2 Netmasks and Routing

Netmasks are used to define the address range of a subnet. If two hosts are in the same subnet, they can reach each other directly. If they are not in the same subnet, they need the address of a gateway that handles all the traffic for the subnet. To check if two IP addresses are in the same subnet, simply “AND” both addresses with the netmask. If the result is identical, both IP addresses are in the same local network. If there are differences, the remote IP address, and thus the remote interface, can only be reached over a gateway.

To understand how the netmask works, look at [Example 15.2, “Linking IP Addresses to the Netmask”](#). The netmask consists of 32 bits that identify how much of an IP address belongs to the network. All those bits that are 1 mark the corresponding bit in the IP address as belonging to the network. All bits that are 0 mark bits inside the subnet. This means that the more bits are 1, the smaller the subnet is. Because the netmask always consists of several successive 1 bits, it is also possible to count the number of bits in the netmask. In [Example 15.2, “Linking IP Addresses to the Netmask”](#) the first net with 24 bits could also be written as 192.168.0.0/24.

EXAMPLE 15.2: LINKING IP ADDRESSES TO THE NETMASK

```
IP address (192.168.0.20): 11000000 10101000 00000000 00010100
Netmask   (255.255.255.0): 11111111 11111111 11111111 00000000
-----
Result of the link:      11000000 10101000 00000000 00000000
In the decimal system:   192.    168.    0.    0

IP address (213.95.15.200): 11010101 10111111 00001111 11001000
Netmask   (255.255.255.0): 11111111 11111111 11111111 00000000
-----
Result of the link:      11010101 10111111 00001111 00000000
In the decimal system:   213.    95.    15.    0
```

To give another example: all machines connected with the same Ethernet cable are usually located in the same subnet and are directly accessible. Even when the subnet is physically divided by switches or bridges, these hosts can still be reached directly.

IP addresses outside the local subnet can only be reached if a gateway is configured for the target network. In the most common case, there is only one gateway that handles all traffic that is external. However, it is also possible to configure several gateways for different subnets.

If a gateway has been configured, all external IP packets are sent to the appropriate gateway. This gateway then attempts to forward the packets in the same manner—from host to host—until it reaches the destination host or the packet's TTL (time to live) expires.

SPECIFIC ADDRESSES

Base Network Address

This is the netmask AND any address in the network, as shown in *Example 15.2, “Linking IP Addresses to the Netmask”* under Result. This address cannot be assigned to any hosts.

Broadcast Address

This could be paraphrased as: “Access all hosts in this subnet.” To generate this, the netmask is inverted in binary form and linked to the base network address with a logical OR. The above example therefore results in 192.168.0.255. This address cannot be assigned to any hosts.

Local Host

The address 127.0.0.1 is assigned to the “loopback device” on each host. A connection can be set up to your own machine with this address and with all addresses from the complete 127.0.0.0/8 loopback network as defined with IPv4. With IPv6 there is only one loopback address (::1).

Because IP addresses must be unique all over the world, you cannot select random addresses. There are three address domains to use if you want to set up a private IP-based network. These cannot get any connection from the rest of the Internet, because they cannot be transmitted over the Internet. These address domains are specified in RFC 1597 and listed in *Table 15.1, “Private IP Address Domains”*.

TABLE 15.1: PRIVATE IP ADDRESS DOMAINS

Network/Netmask	Domain
<u>10.0.0.0 / 255.0.0.0</u>	<u>10.x.x.x</u>
<u>172.16.0.0 / 255.240.0.0</u>	<u>172.16.x.x – 172.31.x.x</u>
<u>192.168.0.0 / 255.255.0.0</u>	<u>192.168.x.x</u>

15.2 IPv6—The Next Generation Internet

Important: IBM z Systems: IPv6 Support

IPv6 is not supported by the CTC and IUCV network connections of the IBM z Systems hardware.

Because of the emergence of the WWW (World Wide Web), the Internet has experienced explosive growth, with an increasing number of computers communicating via TCP/IP in the past fifteen years. Since Tim Berners-Lee at CERN (<http://public.web.cern.ch>) invented the WWW in 1990, the number of Internet hosts has grown from a few thousand to about a hundred million. As mentioned, an IPv4 address consists of only 32 bits. Also, quite a few IP addresses are lost—they cannot be used because of the way in which networks are organized. The number of addresses available in your subnet is two to the power of the number of bits, minus two. A subnet has, for example, 2, 6, or 14 addresses available. To connect 128 hosts to the Internet, for example, you need a subnet with 256 IP addresses, from which only 254 are usable, because two IP addresses are needed for the structure of the subnet itself: the broadcast and the base network address.

Under the current IPv4 protocol, DHCP or NAT (network address translation) are the typical mechanisms used to circumvent the potential address shortage. Combined with the convention to keep private and public address spaces separate, these methods can certainly mitigate the shortage. The problem with them lies in their configuration, which is a chore to set up and a burden to maintain. To set up a host in an IPv4 network, you need several address items, such as the host's own IP address, the subnetmask, the gateway address and maybe a name server address. All these items need to be known and cannot be derived from somewhere else.

With IPv6, both the address shortage and the complicated configuration should be a thing of the past. The following sections tell more about the improvements and benefits brought by IPv6 and about the transition from the old protocol to the new one.

15.2.1 Advantages

The most important and most visible improvement brought by the new protocol is the enormous expansion of the available address space. An IPv6 address is made up of 128 bit values instead of the traditional 32 bits. This provides for as many as several quadrillion IP addresses.

However, IPv6 addresses are not only different from their predecessors with regard to their length. They also have a different internal structure that may contain more specific information about the systems and the networks to which they belong. More details about this are found in [Section 15.2.2, “Address Types and Structure”](#).

The following is a list of other advantages of the new protocol:

Autoconfiguration

IPv6 makes the network “plug and play” capable, which means that a newly set up system integrates into the (local) network without any manual configuration. The new host uses its automatic configuration mechanism to derive its own address from the information made available by the neighboring routers, relying on a protocol called the *neighbor discovery* (ND) protocol. This method does not require any intervention on the administrator's part and there is no need to maintain a central server for address allocation—an additional advantage over IPv4, where automatic address allocation requires a DHCP server.

Nevertheless if a router is connected to a switch, the router should send periodic advertisements with flags telling the hosts of a network how they should interact with each other. For more information, see RFC 2462 and the [radvd.conf\(5\)](#) man page, and RFC 3315.

Mobility

IPv6 makes it possible to assign several addresses to one network interface at the same time. This allows users to access several networks easily, something that could be compared with the international roaming services offered by mobile phone companies: when you take your mobile phone abroad, the phone automatically logs in to a foreign service when it enters the corresponding area, so you can be reached under the same number everywhere and can place an outgoing call, as you would in your home area.

Secure Communication

With IPv4, network security is an add-on function. IPv6 includes IPsec as one of its core features, allowing systems to communicate over a secure tunnel to avoid eavesdropping by outsiders on the Internet.

Backward Compatibility

Realistically, it would be impossible to switch the entire Internet from IPv4 to IPv6 at one time. Therefore, it is crucial that both protocols can coexist not only on the Internet, but also on one system. This is ensured by compatible addresses (IPv4 addresses can easily be translated into IPv6 addresses) and by using several tunnels. See [Section 15.2.3, “Coexistence of IPv4 and IPv6”](#). Also, systems can rely on a *dual stack IP* technique to support both protocols at the same time, meaning that they have two network stacks that are completely separate, such that there is no interference between the two protocol versions.

Custom Tailored Services through Multicasting

With IPv4, some services, such as SMB, need to broadcast their packets to all hosts in the local network. IPv6 allows a much more fine-grained approach by enabling servers to address hosts through *multicasting*—by addressing several hosts as parts of a group (which is different from addressing all hosts through *broadcasting* or each host individually through *unicasting*). Which hosts are addressed as a group may depend on the concrete application. There are some predefined groups to address all name servers (the *all name servers multicast group*), for example, or all routers (the *all routers multicast group*).

15.2.2 Address Types and Structure

As mentioned, the current IP protocol is lacking in two important aspects: there is an increasing shortage of IP addresses and configuring the network and maintaining the routing tables is becoming a more complex and burdensome task. IPv6 solves the first problem by expanding the

address space to 128 bits. The second one is countered by introducing a hierarchical address structure, combined with sophisticated techniques to allocate network addresses, and *multihoming* (the ability to assign several addresses to one device, giving access to several networks).

When dealing with IPv6, it is useful to know about three different types of addresses:

Unicast

Addresses of this type are associated with exactly one network interface. Packets with such an address are delivered to only one destination. Accordingly, unicast addresses are used to transfer packets to individual hosts on the local network or the Internet.

Multicast

Addresses of this type relate to a group of network interfaces. Packets with such an address are delivered to all destinations that belong to the group. Multicast addresses are mainly used by certain network services to communicate with certain groups of hosts in a well-directed manner.

Anycast

Addresses of this type are related to a group of interfaces. Packets with such an address are delivered to the member of the group that is closest to the sender, according to the principles of the underlying routing protocol. Anycast addresses are used to make it easier for hosts to find out about servers offering certain services in the given network area. All servers of the same type have the same anycast address. Whenever a host requests a service, it receives a reply from the server with the closest location, as determined by the routing protocol. If this server should fail for some reason, the protocol automatically selects the second closest server, then the third one, and so forth.

An IPv6 address is made up of eight four-digit fields, each representing 16 bits, written in hexadecimal notation. They are separated by colons (:). Any leading zero bytes within a given field may be dropped, but zeros within the field or at its end may not. Another convention is that more than four consecutive zero bytes may be collapsed into a double colon. However, only one such `::` is allowed per address. This kind of shorthand notation is shown in *Example 15.3, "Sample IPv6 Address"*, where all three lines represent the same address.

EXAMPLE 15.3: SAMPLE IPV6 ADDRESS

```
fe80 : 0000 : 0000 : 0000 : 0000 : 10 : 1000 : 1a4
fe80 :    0 :    0 :    0 :    0 : 10 : 1000 : 1a4
fe80 :                               : 10 : 1000 : 1a4
```

Each part of an IPv6 address has a defined function. The first bytes form the prefix and specify the type of address. The center part is the network portion of the address, but it may be unused. The end of the address forms the host part. With IPv6, the netmask is defined by indicating the length of the prefix after a slash at the end of the address. An address, as shown in *Example 15.4, "IPv6 Address Specifying the Prefix Length"*, contains the information that the first 64 bits form the network part of the address and the last 64 form its host part. In other words, the 64 means that the netmask is filled with 64 1-bit values from the left. As with IPv4, the IP address is combined with AND with the values from the netmask to determine whether the host is located in the same subnet or in another one.

EXAMPLE 15.4: IPV6 ADDRESS SPECIFYING THE PREFIX LENGTH

```
fe80::10:1000:1a4/64
```

IPv6 knows about several predefined types of prefixes. Some are shown in *Various IPv6 Prefixes*.

VARIOUS IPV6 PREFIXES

00

IPv4 addresses and IPv4 over IPv6 compatibility addresses. These are used to maintain compatibility with IPv4. Their use still requires a router able to translate IPv6 packets into IPv4 packets. Several special addresses, such as the one for the loopback device, have this prefix as well.

2 or 3 as the first digit

Aggregatable global unicast addresses. As is the case with IPv4, an interface can be assigned to form part of a certain subnet. Currently, there are the following address spaces: 2001::/16 (production quality address space) and 2002::/16 (6to4 address space).

fe80::/10

Link-local addresses. Addresses with this prefix should not be routed and should therefore only be reachable from within the same subnet.

fec0::/10

Site-local addresses. These may be routed, but only within the network of the organization to which they belong. In effect, they are the IPv6 equivalent of the current private network address space, such as 10.x.x.x.

ff

These are multicast addresses.

A unicast address consists of three basic components:

Public Topology

The first part (which also contains one of the prefixes mentioned above) is used to route packets through the public Internet. It includes information about the company or institution that provides the Internet access.

Site Topology

The second part contains routing information about the subnet to which to deliver the packet.

Interface ID

The third part identifies the interface to which to deliver the packet. This also allows for the MAC to form part of the address. Given that the MAC is a globally unique, fixed identifier coded into the device by the hardware maker, the configuration procedure is substantially simplified. In fact, the first 64 address bits are consolidated to form the EUI-64 token, with the last 48 bits taken from the MAC, and the remaining 24 bits containing special information about the token type. This also makes it possible to assign an EUI-64 token to interfaces that do not have a MAC, such as those based on PPP.

On top of this basic structure, IPv6 distinguishes between five different types of unicast addresses:

:: (unspecified)

This address is used by the host as its source address when the interface is initialized for the first time—when the address cannot yet be determined by other means.

:::1 (loopback)

The address of the loopback device.

IPv4 Compatible Addresses

The IPv6 address is formed by the IPv4 address and a prefix consisting of 96 zero bits. This type of compatibility address is used for tunneling (see [Section 15.2.3, “Coexistence of IPv4 and IPv6”](#)) to allow IPv4 and IPv6 hosts to communicate with others operating in a pure IPv4 environment.

IPv4 Addresses Mapped to IPv6

This type of address specifies a pure IPv4 address in IPv6 notation.

Local Addresses

There are two address types for local use:

link-local

This type of address can only be used in the local subnet. Packets with a source or target address of this type should not be routed to the Internet or other subnets. These addresses contain a special prefix (`fe80::/10`) and the interface ID of the network card, with the middle part consisting of zero bytes. Addresses of this type are used during automatic configuration to communicate with other hosts belonging to the same subnet.

site-local

Packets with this type of address may be routed to other subnets, but not to the wider Internet—they must remain inside the organization's own network. Such addresses are used for intranets and are an equivalent of the private address space defined by IPv4. They contain a special prefix (`fec0::/10`), the interface ID, and a 16 bit field specifying the subnet ID. Again, the rest is filled with zero bytes.

As a completely new feature introduced with IPv6, each network interface normally gets several IP addresses, with the advantage that several networks can be accessed through the same interface. One of these networks can be configured completely automatically using the MAC and a known prefix with the result that all hosts on the local network can be reached when IPv6 is enabled (using the link-local address). With the MAC forming part of it, any IP address used in the world is unique. The only variable parts of the address are those specifying the *site topology* and the *public topology*, depending on the actual network in which the host is currently operating.

For a host to go back and forth between different networks, it needs at least two addresses. One of them, the *home address*, not only contains the interface ID but also an identifier of the home network to which it normally belongs (and the corresponding prefix). The home address is a static address and, as such, it does not normally change. Still, all packets destined to the mobile host can be delivered to it, regardless of whether it operates in the home network or somewhere outside. This is made possible by the completely new features introduced with IPv6, such as *stateless autoconfiguration* and *neighbor discovery*. In addition to its home address, a mobile host gets one or more additional addresses that belong to the foreign networks where it is roaming. These are called *care-of* addresses. The home network has a facility that forwards any packets destined to the host when it is roaming outside. In an IPv6 environment, this task is performed

by the *home agent*, which takes all packets destined to the home address and relays them through a tunnel. On the other hand, those packets destined to the care-of address are directly transferred to the mobile host without any special detours.

15.2.3 Coexistence of IPv4 and IPv6

The migration of all hosts connected to the Internet from IPv4 to IPv6 is a gradual process. Both protocols will coexist for some time to come. The coexistence on one system is guaranteed where there is a *dual stack* implementation of both protocols. That still leaves the question of how an IPv6 enabled host should communicate with an IPv4 host and how IPv6 packets should be transported by the current networks, which are predominantly IPv4 based. The best solutions offer tunneling and compatibility addresses (see [Section 15.2.2, “Address Types and Structure”](#)).

IPv6 hosts that are more or less isolated in the (worldwide) IPv4 network can communicate through tunnels: IPv6 packets are encapsulated as IPv4 packets to move them across an IPv4 network. Such a connection between two IPv4 hosts is called a *tunnel*. To achieve this, packets must include the IPv6 destination address (or the corresponding prefix) and the IPv4 address of the remote host at the receiving end of the tunnel. A basic tunnel can be configured manually according to an agreement between the hosts' administrators. This is also called *static tunneling*. However, the configuration and maintenance of static tunnels is often too labor-intensive to use them for daily communication needs. Therefore, IPv6 provides for three different methods of *dynamic tunneling*:

6over4

IPv6 packets are automatically encapsulated as IPv4 packets and sent over an IPv4 network capable of multicasting. IPv6 is tricked into seeing the whole network (Internet) as a huge local area network (LAN). This makes it possible to determine the receiving end of the IPv4 tunnel automatically. However, this method does not scale very well and is also hampered because IP multicasting is far from widespread on the Internet. Therefore, it only provides a solution for smaller corporate or institutional networks where multicasting can be enabled. The specifications for this method are laid down in RFC 2529.

6to4

With this method, IPv4 addresses are automatically generated from IPv6 addresses, enabling isolated IPv6 hosts to communicate over an IPv4 network. However, several problems have been reported regarding the communication between those isolated IPv6 hosts and the Internet. The method is described in RFC 3056.

IPv6 Tunnel Broker

This method relies on special servers that provide dedicated tunnels for IPv6 hosts. It is described in RFC 3053.

15.2.4 Configuring IPv6

To configure IPv6, you normally do not need to make any changes on the individual workstations. IPv6 is enabled by default. To disable or enable IPv6 on an installed system, use the YaST *Network Settings* module. On the *Global Options* tab, check or uncheck the *Enable IPv6* option as necessary. If you want to enable it temporarily until the next reboot, enter `modprobe -i ipv6` as `root`. It is impossible to unload the IPv6 module after it has been loaded.

Because of the autoconfiguration concept of IPv6, the network card is assigned an address in the *link-local* network. Normally, no routing table management takes place on a workstation. The network routers can be queried by the workstation, using the *router advertisement protocol*, for what prefix and gateways should be implemented. The `radvd` program can be used to set up an IPv6 router. This program informs the workstations which prefix to use for the IPv6 addresses and which routers. Alternatively, use `zebra/quagga` for automatic configuration of both addresses and routing.

For information about how to set up various types of tunnels using the `/etc/sysconfig/network` files, see the man page of `ifcfg-tunnel` (`man ifcfg-tunnel`).

15.2.5 For More Information

The above overview does not cover the topic of IPv6 comprehensively. For a more in-depth look at the new protocol, refer to the following online documentation and books:

<http://www.ipv6.org/> ↗

The starting point for everything about IPv6.

<http://www.ipv6day.org> ↗

All information needed to start your own IPv6 network.

<http://www.ipv6-to-standard.org/> ↗

The list of IPv6-enabled products.

<http://www.bieringer.de/linux/IPv6/> ↗

Here, find the Linux IPv6-HOWTO and many links related to the topic.

The fundamental RFC about IPv6.

IPv6 Essentials

A book describing all the important aspects of the topic is *IPv6 Essentials* by Silvia Hagen (ISBN 0-596-00125-8).


15.3 Name Resolution

DNS assists in assigning an IP address to one or more names and assigning a name to an IP address. In Linux, this conversion is usually carried out by a special type of software known as *bind*. The machine that takes care of this conversion is called a *name server*. The names make up a hierarchical system in which each name component is separated by a period. The name hierarchy is, however, independent of the IP address hierarchy described above.

Consider a complete name, such as jupiter.example.com, written in the format hostname.domain. A full name, called a *fully qualified domain name* (FQDN), consists of a host name and a domain name (example.com). The latter also includes the *top level domain* or TLD (com).

TLD assignment has become quite confusing for historical reasons. Traditionally, three-letter domain names are used in the USA. In the rest of the world, the two-letter ISO national codes are the standard. In addition to that, longer TLDs were introduced in 2000 that represent certain spheres of activity (for example, .info, .name, .museum).

In the early days of the Internet (before 1990), the file /etc/hosts was used to store the names of all the machines represented over the Internet. This quickly proved to be impractical in the face of the rapidly growing number of computers connected to the Internet. For this reason, a decentralized database was developed to store the host names in a widely distributed manner. This database, similar to the name server, does not have the data pertaining to all hosts in the Internet readily available, but can dispatch requests to other name servers.

The top of the hierarchy is occupied by *root name servers*. These root name servers manage the top level domains and are run by the Network Information Center (NIC). Each root name server knows about the name servers responsible for a given top level domain. Information about top level domain NICs is available at <http://www.internic.net> .

DNS can do more than resolve host names. The name server also knows which host is receiving e-mails for an entire domain—the *mail exchanger* (MX).

For your machine to resolve an IP address, it must know about at least one name server and its IP address. Easily specify such a name server using YaST. The configuration of name server access with SUSE® Linux Enterprise Server is described in *Section 15.4.1.4, “Configuring Host Name and DNS”*. Setting up your own name server is described in *Chapter 23, The Domain Name System*. The protocol `whois` is closely related to DNS. With this program, quickly find out who is responsible for a given domain.



Note: MDNS and .local Domain Names

The `.local` top level domain is treated as link-local domain by the resolver. DNS requests are sent as multicast DNS requests instead of normal DNS requests. If you already use the `.local` domain in your name server configuration, you must switch this option off in `/etc/host.conf`. For more information, see the `host.conf` manual page.

If you want to switch off MDNS during installation, use `nomdns=1` as a boot parameter.

For more information on multicast DNS, see <http://www.multicastdns.org>.

15.4 Configuring a Network Connection with YaST

There are many supported networking types on Linux. Most of them use different device names and the configuration files are spread over several locations in the file system. For a detailed overview of the aspects of manual network configuration, see *Section 15.5, “Configuring a Network Connection Manually”*.

All network interfaces with link up (with a network cable connected) are automatically configured. Additional hardware can be configured any time on the installed system. The following sections describe the network configuration for all types of network connections supported by SUSE Linux Enterprise Server.



Tip: IBM z Systems: Hotpluggable Network Cards

On IBM z Systems platforms, hotpluggable network cards are supported, but not their automatic network integration via DHCP (as is the case on the PC). After detection, manually configure the interface.

15.4.1 Configuring the Network Card with YaST

To configure your Ethernet or Wi-Fi/Bluetooth card in YaST, select *System > Network Settings*. After starting the module, YaST displays the *Network Settings* dialog with four tabs: *Global Options*, *Overview*, *Hostname/DNS* and *Routing*.

The *Global Options* tab allows you to set general networking options such as the network setup method, IPv6, and general DHCP options. For more information, see [Section 15.4.1.1, “Configuring Global Networking Options”](#).

The *Overview* tab contains information about installed network interfaces and configurations. Any properly detected network card is listed with its name. You can manually configure new cards, remove or change their configuration in this dialog. If you want to manually configure a card that was not automatically detected, see [Section 15.4.1.3, “Configuring an Undetected Network Card”](#). If you want to change the configuration of an already configured card, see [Section 15.4.1.2, “Changing the Configuration of a Network Card”](#).

The *Hostname/DNS* tab allows to set the host name of the machine and name the servers to be used. For more information, see [Section 15.4.1.4, “Configuring Host Name and DNS”](#).

The *Routing* tab is used for the configuration of routing. See [Section 15.4.1.5, “Configuring Routing”](#) for more information.

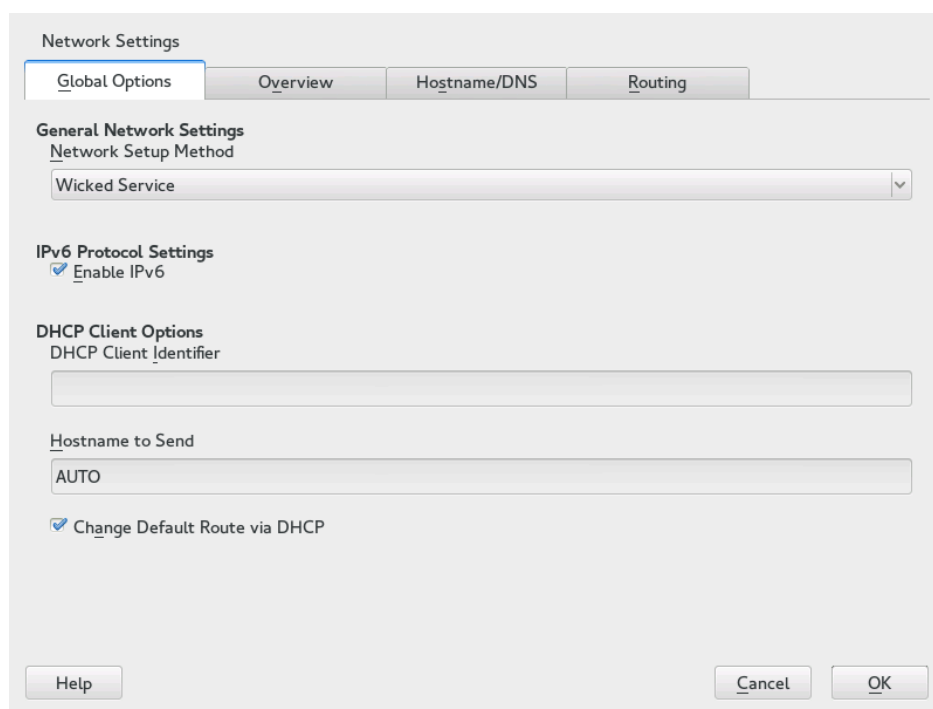


FIGURE 15.3: CONFIGURING NETWORK SETTINGS

15.4.1.1 Configuring Global Networking Options

The *Global Options* tab of the YaST *Network Settings* module allows you to set important global networking options, such as the use of NetworkManager, IPv6 and DHCP client options. These settings are applicable for all network interfaces.



Note: NetworkManager Provided by Workstation Extension

NetworkManager is now provided by the Workstation Extension. To install NetworkManager, activate the Workstation Extension repository, and select the NetworkManager packages.

In the *Network Setup Method* choose the way network connections are managed. If you want a NetworkManager desktop applet to manage connections for all interfaces, choose *NetworkManager Service*. NetworkManager is well suited for switching between multiple wired and wireless networks. If you do not run a desktop environment, or if your computer is a Xen server, virtual system, or provides network services such as DHCP or DNS in your network, use the *Wicked Service* method. If NetworkManager is used, **nm-applet** should be used to configure network options and the *Overview*, *Hostname/DNS* and *Routing* tabs of the *Network Settings* module are disabled. For more information on NetworkManager, see the SUSE Linux Enterprise Desktop documentation.

In the *IPv6 Protocol Settings* choose whether to use the IPv6 protocol. It is possible to use IPv6 together with IPv4. By default, IPv6 is enabled. However, in networks not using IPv6 protocol, response times can be faster with IPv6 protocol disabled. To disable IPv6, deactivate *Enable IPv6*. If IPv6 is disabled, the Kernel no longer loads the IPv6 module automatically. This setting will be applied after reboot.

In the *DHCP Client Options* configure options for the DHCP client. The *DHCP Client Identifier* must be different for each DHCP client on a single network. If left empty, it defaults to the hardware address of the network interface. However, if you are running several virtual machines using the same network interface and, therefore, the same hardware address, specify a unique free-form identifier here.

The *Hostname to Send* specifies a string used for the host name option field when the DHCP client sends messages to DHCP server. Some DHCP servers update name server zones (forward and reverse records) according to this host name (Dynamic DNS). Also, some DHCP servers require

the *Hostname to Send* option field to contain a specific string in the DHCP messages from clients. Leave AUTO to send the current host name (that is the one defined in /etc/HOSTNAME). Make the option field empty for not sending any host name.

If you do not want to change the default route according to the information from DHCP, deactivate *Change Default Route via DHCP*.

15.4.1.2 Changing the Configuration of a Network Card

To change the configuration of a network card, select a card from the list of the detected cards in *Network Settings > Overview* in YaST and click *Edit*. The *Network Card Setup* dialog appears in which to adjust the card configuration using the *General*, *Address* and *Hardware* tabs.

15.4.1.2.1 Configuring IP Addresses

You can set the IP address of the network card or the way its IP address is determined in the *Address* tab of the *Network Card Setup* dialog. Both IPv4 and IPv6 addresses are supported. The network card can have *No IP Address* (which is useful for bonding devices), a *Statically Assigned IP Address* (IPv4 or IPv6) or a *Dynamic Address* assigned via *DHCP* or *Zeroconf* or both.

If using *Dynamic Address*, select whether to use *DHCP Version 4 Only* (for IPv4), *DHCP Version 6 Only* (for IPv6) or *DHCP Both Version 4 and 6*.

If possible, the first network card with link that is available during the installation is automatically configured to use automatic address setup via DHCP.



Note: IBM z Systems and DHCP

On IBM z Systems platforms, DHCP-based address configuration is only supported with network cards that have a MAC address. This is only the case with OSA and OSA Express cards.

DHCP should also be used if you are using a DSL line but with no static IP assigned by the ISP (Internet Service Provider). If you decide to use DHCP, configure the details in *DHCP Client Options* in the *Global Options* tab of the *Network Settings* dialog of the YaST network card configuration module. If you have a virtual host setup where different hosts communicate through the same interface, an *DHCP Client Identifier* is necessary to distinguish them.

DHCP is a good choice for client configuration but it is not ideal for server configuration. To set a static IP address, proceed as follows:

1. Select a card from the list of detected cards in the *Overview* tab of the YaST network card configuration module and click *Edit*.
2. In the *Address* tab, choose *Statically Assigned IP Address*.
3. Enter the *IP Address*. Both IPv4 and IPv6 addresses can be used. Enter the network mask in *Subnet Mask*. If the IPv6 address is used, use *Subnet Mask* for prefix length in format */64*. Optionally, you can enter a fully qualified *Hostname* for this address, which will be written to the */etc/hosts* configuration file.
4. Click *Next*.
5. To activate the configuration, click *OK*.

If you use the static address, the name servers and default gateway are not configured automatically. To configure name servers, proceed as described in [Section 15.4.1.4, “Configuring Host Name and DNS”](#). To configure a gateway, proceed as described in [Section 15.4.1.5, “Configuring Routing”](#).

15.4.1.2.2 Configuring Multiple Addresses

One network device can have multiple IP addresses.



Note: Aliases Are a Compatibility Feature

These so-called aliases or labels, respectively, work with IPv4 only. With IPv6 they will be ignored. Using **iproute2** network interfaces can have one or more addresses.

Using YaST to set additional addresses for your network card, proceed as follows:

1. Select a card from the list of detected cards in the *Overview* tab of the YaST *Network Settings* dialog and click *Edit*.
2. In the *Address > Additional Addresses* tab, click *Add*.
3. Enter *IPv4 Address Label*, *IP Address*, and *Netmask*. Do not include the interface name in the alias name.

4. To activate the configuration, confirm the settings.

15.4.1.2.3 Changing the Device Name and Udev Rules

It is possible to change the device name of the network card when it is used. It is also possible to determine whether the network card should be identified by udev via its hardware (MAC) address or via the bus ID. The later option is preferable in large servers to simplify hotplugging of cards. To set these options with YaST, proceed as follows:

1. Select a card from the list of detected cards in the *Overview* tab of the YaST *Network Settings* dialog and click *Edit*.
2. Go to the *Hardware* tab. The current device name is shown in *Udev Rules*. Click *Change*.
3. Select whether udev should identify the card by its *MAC Address* or *Bus ID*. The current MAC address and bus ID of the card are shown in the dialog.
4. To change the device name, check the *Change Device Name* option and edit the name.
5. To activate the configuration, confirm the settings.

15.4.1.2.4 Changing Network Card Kernel Driver

For some network cards, several Kernel drivers may be available. If the card is already configured, YaST allows you to select a Kernel driver to be used from a list of available suitable drivers. It is also possible to specify options for the Kernel driver. To set these options with YaST, proceed as follows:

1. Select a card from the list of detected cards in the *Overview* tab of the YaST *Network Settings* module and click *Edit*.
2. Go to the *Hardware* tab.
3. Select the Kernel driver to be used in *Module Name*. Enter any options for the selected driver in *Options* in the form `= = value`. If more options are used, they should be space-separated.
4. To activate the configuration, confirm the settings.

15.4.1.2.5 Activating the Network Device

If you use the method with wicked, you can configure your device to either start during boot, on cable connection, on card detection, manually, or never. To change device start-up, proceed as follows:

1. In YaST select a card from the list of detected cards in *System > Network Settings* and click *Edit*.
2. In the *General* tab, select the desired entry from *Device Activation*.
Choose *At Boot Time* to start the device during the system boot. With *On Cable Connection*, the interface is watched for any existing physical connection. With *On Hotplug*, the interface is set when available. It is similar to the *At Boot Time* option, and only differs in that no error occurs if the interface is not present at boot time. Choose *Manually* to control the interface manually with ifup. Choose *Never* to not start the device. The *On NFSroot* is similar to *At Boot Time*, but the interface does not shut down with the systemctl stop network command; the network service also cares about the wicked service if wicked is active. Use this if you use an NFS or iSCSI root file system.
3. To activate the configuration, confirm the settings.



Tip: NFS as a Root File System

On (diskless) systems where the root partition is mounted via network as an NFS share, you need to be careful when configuring the network device with which the NFS share is accessible.

When shutting down or rebooting the system, the default processing order is to turn off network connections, then unmount the root partition. With NFS root, this order causes problems as the root partition cannot be cleanly unmounted as the network connection to the NFS share is already not activated. To prevent the system from deactivating the relevant network device, open the network device configuration tab as described in [Section 15.4.1.2.5, “Activating the Network Device”](#), and choose *On NFSroot* in the *Device Activation* pane.

15.4.1.2.6 Setting Up Maximum Transfer Unit Size

You can set a maximum transmission unit (MTU) for the interface. MTU refers to the largest allowed packet size in bytes. A higher MTU brings higher bandwidth efficiency. However, large packets can block up a slow interface for some time, increasing the lag for further packets.

1. In YaST select a card from the list of detected cards in *System > Network Settings* and click *Edit*.
2. In the *General* tab, select the desired entry from the *Set MTU* list.
3. To activate the configuration, confirm the settings.

15.4.1.2.7 PCIe Multifunction Devices

Multifunction devices that support LAN, iSCSI, and FCoE are supported. YaST FCoE client (**yast2 fcoe-client**) shows the private flags in additional columns to allow the user to select the device meant for FCoE. YaST network module (**yast2 lan**) excludes “storage only devices” for network configuration.

For more information about FCoE, see *Book “Storage Administration Guide”, Chapter 15 “Fibre Channel Storage over Ethernet Networks: FCoE”, Section 15.3 “Managing FCoE Services with YaST”*.

15.4.1.2.8 Infiniband Configuration for IP-over-InfiniBand (IPoIB)

1. In YaST select the InfiniBand device in *System > Network Settings* and click *Edit*.
2. In the *General* tab, select one of the *IP-over-InfiniBand* (IPoIB) modes: *connected* (default) or *datagram*.
3. To activate the configuration, confirm the settings.

For more information about InfiniBand, see </usr/src/linux/Documentation/infiniband/ipoib.txt>.

15.4.1.2.9 Configuring the Firewall

Without having to enter the detailed firewall setup as described in *Book “Security Guide”, Chapter 15 “Masquerading and Firewalls”, Section 15.4.1 “Configuring the Firewall with YaST”*, you can determine the basic firewall setup for your device as part of the device setup. Proceed as follows:

1. Open the YaST *System > Network Settings* module. In the *Overview* tab, select a card from the list of detected cards and click *Edit*.
2. Enter the *General* tab of the *Network Settings* dialog.
3. Determine the *Firewall Zone* to which your interface should be assigned. The following options are available:

Firewall Disabled

This option is available only if the firewall is disabled and the firewall does not run. Only use this option if your machine is part of a greater network that is protected by an outer firewall.

Automatically Assign Zone

This option is available only if the firewall is enabled. The firewall is running and the interface is automatically assigned to a firewall zone. The zone which contains the keyword any or the external zone will be used for such an interface.

Internal Zone (Unprotected)

The firewall is running, but does not enforce any rules to protect this interface. Use this option if your machine is part of a greater network that is protected by an outer firewall. It is also useful for the interfaces connected to the internal network, when the machine has more network interfaces.

Demilitarized Zone

A demilitarized zone is an additional line of defense in front of an internal network and the (hostile) Internet. Hosts assigned to this zone can be reached from the internal network and from the Internet, but cannot access the internal network.

External Zone

The firewall is running on this interface and fully protects it against other—presumably hostile—network traffic. This is the default option.

4. To activate the configuration, confirm the settings.

15.4.1.3 Configuring an Undetected Network Card

If a network card is not detected correctly, the card is not included in the list of detected cards. If you are sure that your system includes a driver for your card, you can configure it manually. You can also configure special network device types, such as bridge, bond, TUN or TAP. To configure an undetected network card (or a special device) proceed as follows:

1. In the *System > Network Settings > Overview* dialog in YaST click *Add*.
2. In the *Hardware* dialog, set the *Device Type* of the interface from the available options and *Configuration Name*. If the network card is a PCMCIA or USB device, activate the respective check box and exit this dialog with *Next*. Otherwise, you can define the *Kernel Module Name* to be used for the card and its *Options*, if necessary.
In *Ethtool Options*, you can set **ethtool** options used by **ifup** for the interface. For information about available options, see the **ethtool** manual page.
If the option string starts with a `-` (for example, `-K interface_name rx on`), the second word in the string is replaced with the current interface name. Otherwise (for example, `autoneg off speed 10`) **ifup** adds `-s interface_name` to the beginning.
3. Click *Next*.
4. Configure any needed options, such as the IP address, device activation or firewall zone for the interface in the *General*, *Address*, and *Hardware* tabs. For more information about the configuration options, see [Section 15.4.1.2, "Changing the Configuration of a Network Card"](#).
5. If you selected *Wireless* as the device type of the interface, configure the wireless connection in the next dialog.
6. To activate the new network configuration, confirm the settings.

15.4.1.4 Configuring Host Name and DNS

If you did not change the network configuration during installation and the Ethernet card was already available, a host name was automatically generated for your computer and DHCP was activated. The same applies to the name service information your host needs to integrate into a network environment. If DHCP is used for network address setup, the list of domain name servers is automatically filled with the appropriate data. If a static setup is preferred, set these values manually.

To change the name of your computer and adjust the name server search list, proceed as follows:

1. Go to the *Network Settings* > *Hostname/DNS* tab in the *System* module in YaST.
2. Enter the *Hostname* and, if needed, the *Domain Name*. The domain is especially important if the machine is a mail server. Note that the host name is global and applies to all set network interfaces.

If you are using DHCP to get an IP address, the host name of your computer will be automatically set by the DHCP. You should disable this behavior if you connect to different networks, because they may assign different host names and changing the host name at runtime may confuse the graphical desktop. To disable using DHCP to get an IP address deactivate *Change Hostname via DHCP*.

Assign Hostname to Loopback IP associates your host name with 127.0.0.2 (loopback) IP address in /etc/hosts. This is a useful option if you want to have the host name resolvable at all times, even without active network.

3. In *Modify DNS Configuration*, select the way the DNS configuration (name servers, search list, the content of the /etc/resolv.conf file) is modified.

If the *Use Default Policy* option is selected, the configuration is handled by the netconfig script which merges the data defined statically (with YaST or in the configuration files) with data obtained dynamically (from the DHCP client or NetworkManager). This default policy is usually sufficient.

If the *Only Manually* option is selected, netconfig is not allowed to modify the /etc/resolv.conf file. However, this file can be edited manually.

If the *Custom Policy* option is selected, a *Custom Policy Rule* string defining the merge policy should be specified. The string consists of a comma-separated list of interface names to be considered a valid source of settings. Except for complete interface names, basic wild cards to match multiple interfaces are allowed, as well. For example, eth* ppp? will first target all eth and then all ppp0-ppp9 interfaces. There are two special policy values that indicate how to apply the static settings defined in the /etc/sysconfig/network/config file:

STATIC

The static settings need to be merged together with the dynamic settings.

STATIC_FALLBACK

The static settings are used only when no dynamic configuration is available.

For more information, see the man page of netconfig(8) (man 8 netconfig).

4. Enter the *Name Servers* and fill in the *Domain Search* list. Name servers must be specified by IP addresses, such as 192.168.1.116, not by host names. Names specified in the *Domain Search* tab are domain names used for resolving host names without a specified domain. If more than one *Domain Search* is used, separate domains with commas or white space.
5. To activate the configuration, confirm the settings.

It is also possible to edit the host name using YaST from the command line. The changes made by YaST take effect immediately (which is not the case when editing the `/etc/HOSTNAME` file manually). To change the host name, use the following command:

```
yast dns edit hostname=hostname
```

To change the name servers, use the following commands:

```
yast dns edit nameserver1=192.168.1.116  
yast dns edit nameserver2=192.168.1.117  
yast dns edit nameserver3=192.168.1.118
```

15.4.1.5 Configuring Routing

To make your machine communicate with other machines and other networks, routing information must be given to make network traffic take the correct path. If DHCP is used, this information is automatically provided. If a static setup is used, this data must be added manually.

1. In YaST go to *Network Settings > Routing*.
2. Enter the IP address of the *Default Gateway* (IPv4 and IPv6 if necessary). The default gateway matches every possible destination, but if a routing table entry exists that matches the required address, this will be used instead of the default route via the Default Gateway.
3. More entries can be entered in the *Routing Table*. Enter the *Destination* network IP address, *Gateway* IP address and the *Netmask*. Select the *Device* through which the traffic to the defined network will be routed (the minus sign stands for any device). To omit any of these values, use the minus sign `-`. To enter a default gateway into the table, use `default` in the *Destination* field.



Note: Route Prioritization

If more default routes are used, it is possible to specify the metric option to determine which route has a higher priority. To specify the metric option, enter _metric number in *Options*. The route with the highest metric is used as default. If the network device is disconnected, its route will be removed and the next one will be used. However, the current Kernel does not use metric in static routing, only routing daemons like multipathd do.

4. If the system is a router, enable *IPv4 Forwarding* and *IPv6 Forwarding* in the *Network Settings* as needed.
5. To activate the configuration, confirm the settings.

15.4.2 IBM z Systems: Configuring Network Devices

SUSE Linux Enterprise Server for IBM z Systems supports several types of network interfaces. YaST can be used to configure all of them.

15.4.2.1 The qeth-hsi Device

To add a qeth-hsi (Hipersockets) interface to the installed system, start the *System > Network Settings* module in YaST. Select one of the devices marked *Hipersocket* to use as the READ device address and click *Edit*. Enter the device numbers for the read, write and control channels (example device number format: 0.0.0800). Then click next. In the *Network Address Setup* dialog, specify the IP address and netmask for the new interface and leave the network configuration by pressing *Next* and *OK*.

15.4.2.2 The qeth-ethernet Device

To add a qeth-ethernet (IBM OSA Express Ethernet Card) interface to the installed system, start the *System > Network Settings* module in YaST. Select one of the devices marked *IBM OSA Express Ethernet Card* to use as the READ device address and click *Edit*. Enter a device number for the read, write and control channels (example device number format: 0.0.0700). Enter the

needed port name, port number (if applicable) and some additional options (see the *Linux for IBM z Systems: Device Drivers, Features, and Commands* manual for reference, http://www.ibm.com/developerworks/linux/linux390/documentation_suse.html), your IP address, and an appropriate netmask. Leave the network configuration with *Next* and *OK*.

15.4.2.3 The `ctc` Device

To add a `ctc` (IBM parallel CTC Adapter) interface to the installed system, start the *System > Network Settings* module in YaST. Select one of the devices marked *IBM Parallel CTC Adapter* to use as your read channel and click *Configure*. Choose the *Device Settings* that fit your devices (usually this would be *Compatibility Mode*). Specify both your IP address and the IP address of the remote partner. If needed, adjust the MTU size with *Advanced > Detailed Settings*. Leave the network configuration with *Next* and *OK*.



Warning: CTC is no Longer Supported

The use of this interface is deprecated. This interface will not be supported in future versions of SUSE Linux Enterprise Server.

15.4.2.4 The `lcs` Device

To add an `lcs` (IBM OSA-2 Adapter) interface to the installed system, start the *System > Network Settings* module in YaST. Select one of the devices marked *IBM OSA-2 Adapter* and click *Configure*. Enter the needed port number, some additional options (see the *Linux for IBM z Systems: Device Drivers, Features, and Commands* manual for reference, http://www.ibm.com/developerworks/linux/linux390/documentation_suse.html), your IP address and an appropriate netmask. Leave the network configuration with *Next* and *OK*.

15.4.2.5 The `IUCV` Device

To add an `iucv` (IUCV) interface to the installed system, start the *System > Network Settings* module in YaST. Select a device marked *IUCV* and click *Edit*. YaST prompts you for the name of your IUCV partner (*Peer*). Enter the name (this entry is case-sensitive) and select *Next*. Specify both the *IP Address* and the *Remote IP Address* of your partner. If needed, *Set MTU* size on *General* tab. Leave the network configuration with *Next* and *OK*.



Warning: IUCV is no Longer Supported

The use of this interface is deprecated. This interface will not be supported in future versions of SUSE Linux Enterprise Server.

15.5 Configuring a Network Connection Manually

Manual configuration of the network software should be the last alternative. Using YaST is recommended. However, this background information about the network configuration can also assist your work with YaST.

15.5.1 The **wicked** Network Configuration

The tool and library called **wicked** provides a new framework for network configuration.

One of the challenges with traditional network interface management is that different layers of network management get jumbled together into one single script, or at most two different scripts, that interact with each other in a not-really-well-defined way, with side effects that are difficult to be aware of, obscure constraints and conventions, etc. Several layers of special hacks for a variety of different scenarios increase the maintenance burden. Address configuration protocols are being used that are implemented via daemons like `dhcpcd`, which interact rather poorly with the rest of the infrastructure. Funky interface naming schemes that require heavy `udev` support are introduced to achieve persistent identification of interfaces.

The idea of `wicked` is to decompose the problem in several ways. None of them is entirely novel, but trying to put ideas from different projects together is hopefully going to create a better solution overall.

One approach is to use a client/server model. This allows `wicked` to define standardized facilities for things like address configuration that are well integrated with the overall framework. For example, with address configuration, the administrator may request that an interface should be configured via DHCP or IPv4 zeroconf, and all the address configuration service does is obtain the lease from its server, and pass it on to the `wicked` server process, which installs the requested addresses and routes.

The other approach to decomposing the problem is to enforce the layering aspect. For any type of network interface, it is possible to define a dbus service that configures the network interface's device layer—a VLAN, a bridge, a bonding, or a paravirtualized device. Common functionality, such as address configuration, is implemented by joint services that are layered on top of these device specific services, without having to implement them specifically.

The wicked framework implements these two aspects by using a variety of dbus services, which get attached to a network interface depending on its type. Here is a rough overview of the current object hierarchy in wicked.

Each network interface is represented via a child object of /org/opensuse/Network/Interfaces. The name of the child object is given by its ifindex. For example, the loopback interface, which usually gets ifindex 1, is /org/opensuse/Network/Interfaces/1, the first Ethernet interface registered is /org/opensuse/Network/Interfaces/2.

Each network interface has a “class” associated with it, which is used to select the dbus interfaces it supports. By default, each network interface is of class netif, and wickedd will automatically attach all interfaces compatible with this class. In the current implementation, this includes the following interfaces:

org.opensuse.Network.Interface

Generic network interface functions, such as taking the link up or down, assigning an MTU, etc.

org.opensuse.Network.Addrconf.ipv4.dhcp,
org.opensuse.Network.Addrconf.ipv6.dhcp,
org.opensuse.Network.Addrconf.ipv4.auto

Address configuration services for DHCP, IPv4 zeroconf, etc.

Beyond this, network interfaces may require or offer special configuration mechanisms. For example, for an Ethernet device, you should be able to control the link speed, offloading of checksumming, etc. To achieve this, Ethernet devices have a class of their own, called netif-ethernet, which is a subclass of netif. As a consequence, the dbus interfaces assigned to an Ethernet interface include all the services listed above, plus org.opensuse.Network.Ethernet, which is a service available only to objects belonging to the netif-ethernet class.

Similarly, there exist classes for interface types like bridges, VLANs, bonds, or infinibands.

How do you interact with an interface that needs to be created first—such as a VLAN, which is really a virtual network interface that sits on top of an Ethernet device. For these, `wicked` defines factory interfaces, such as `org.opensuse.Network.VLAN.Factory`. Such a factory interface offers a single function that lets you create an interface of the requested type. These factory interfaces are attached to the `/org/opensuse/Network/Interfaces` list node.

15.5.1.1 `wicked` Architecture and Features

The `wicked` service comprises several parts as depicted in *Figure 15.4, “wicked architecture”*.

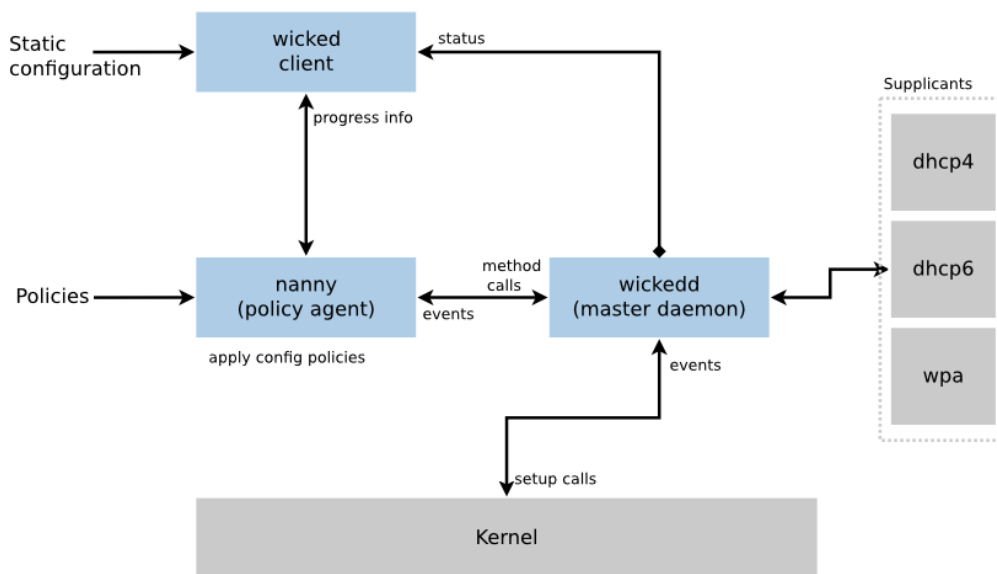


FIGURE 15.4: `wicked` ARCHITECTURE

`wicked` currently supports the following:

- Configuration file back-ends to parse SUSE style `/etc/sysconfig/network` files.
- An internal configuration back-end to represent network interface configuration in XML.
- Bring up and shutdown of “normal” network interfaces such as Ethernet or InfiniBand, VLAN, bridge, bonds, tun, tap, dummy, macvlan, macvtap, hsi, qeth, iucv, and wireless (currently limited to one wpa-psk/eap network) devices.
- A built-in DHCPv4 client and a built-in DHCPv6 client.

- The [Section 15.5.1.3, “Nanny”](#) daemon (enabled by default) helps to automatically bring up configured interfaces when the device is available (interface hotplugging) and set up the IP configuration when a link (carrier) is detected.
- wicked was implemented as a group of DBus services that are integrated with systemd. So the usual **systemctl** commands will apply to wicked.

15.5.1.2 Using wicked

On SUSE Linux Enterprise, wicked is running by default. In case you want to check what is currently enabled and whether it is running, call:

```
systemctl status network
```

If wicked is enabled, you will see something along these lines:

```
wicked.service - wicked managed network interfaces
  Loaded: loaded (/usr/lib/systemd/system/wicked.service; enabled)
  ...
```

In case something different is running (for example, NetworkManager) and you want to switch to wicked, first stop what is running and then enable wicked:

```
systemctl is-active network && \
systemctl stop      network
systemctl enable --force wicked
```

This enables the wicked services, creates the network.service to wicked.service alias link, and starts the network at the next boot.

Starting the server process:

```
systemctl start wickedd
```

This starts wickedd (the main server) and associated supplicants:

```
/usr/lib/wicked/bin/wickedd-auto4 --systemd --foreground
/usr/lib/wicked/bin/wickedd-dhcp4 --systemd --foreground
/usr/lib/wicked/bin/wickedd-dhcp6 --systemd --foreground
/usr/sbin/wickedd --systemd --foreground
/usr/sbin/wickedd-nanny --systemd --foreground
```

Then bringing up the network:

```
systemctl start wicked
```

Alternatively use the `network.service` alias:

```
systemctl start network
```

These commands are using the default or system configuration sources as defined in `/etc/wicked/client.xml`.

To enable debugging, set `WICKED_DEBUG` in `/etc/sysconfig/network/config`, for example:

```
WICKED_DEBUG="all"
```

Or, to omit some:

```
WICKED_DEBUG="all,-dbus,-objectmodel,-xpath,-xml"
```

Use the client utility to display interface information for all interfaces or the interface specified with `ifname`:

```
wicked show all  
wicked show ifname
```

In XML output:

```
wicked show-xml all  
wicked show-xml ifname
```

Bringing up one interface:

```
wicked ifup eth0  
wicked ifup wlan0  
...
```

Because there is no configuration source specified, the wicked client checks its default sources of configuration defined in `/etc/wicked/client.xml`:

1. `firmware`: iSCSI Boot Firmware Table (iBFT)
2. `compat`: `ifcfg` files—implemented for compatibility

Whatever wicked gets from those sources for a given interface is applied. The intended order of importance is firmware, then compat—this may be changed in the future.

For more information, see the wicked man page.

15.5.1.3 Nanny

Nanny is an event and policy driven daemon that is responsible for asynchronous or unsolicited scenarios such as hotplugging devices. Thus the nanny daemon helps with starting or restarting delayed or temporarily gone devices. Nanny monitors device and link changes, and integrates new devices defined by the current policy set. Nanny continues to set up even if ifup already exited because of specified timeout constraints.

By default, the nanny daemon is active on the system. It is enabled in the /etc/wicked/common.xml configuration file:

```
<config>
...
<use-nanny>true</use-nanny>
</config>
```

This setting causes ifup and ifreload to apply a policy with the effective configuration to the nanny daemon; then, nanny configures wickedd and thus ensures hotplug support. It waits in the background for events or changes (such as new devices or carrier on).

15.5.1.4 Bringing Up Multiple Interfaces

For bonds and bridges, it may make sense to define the entire device topology in one file (ifcfg-bondX), and bring it up in one go. wicked then can bring up the whole configuration if you specify the top level interface names (of the bridge or bond):

```
wicked ifup br0
```

This command automatically sets up the bridge and its dependencies in the appropriate order without the need to list the dependencies (ports, etc.) separately.

To bring up multiple interfaces in one command:

```
wicked ifup bond0 br0 br1 br2
```

Or also all interfaces:

```
wicked ifup all
```

15.5.1.5 Using Tunnels with Wicked

When you need to use tunnels with Wicked, the `TUNNEL_DEVICE` is utilized for this. It permits to specify an optional device name to bind the tunnel to the device. The tunneled packets will only be routed via this device.

For more information, refer to [`man 5 ifcfg-tunnel`](#).

15.5.1.6 Handling Incremental Changes

With **wicked**, there is no need to actually take down an interface to reconfigure it (unless it is required by the Kernel). For example, to add another IP address or route to a statically configured network interface, add the IP address to the interface definition, and do another “ifup” operation. The server will try hard to update only those settings that have changed. This applies to link-level options such as the device MTU or the MAC address, and network-level settings, such as addresses, routes, or even the address configuration mode (for example, when moving from a static configuration to DHCP).

Things get tricky of course with virtual interfaces combining several real devices such as bridges or bonds. For bonded devices, it is not possible to change certain parameters while the device is up. Doing that will result in an error.

However, what should still work, is the act of adding or removing the child devices of a bond or bridge, or choosing a bond's primary interface.

15.5.1.7 Wicked Extensions: Address Configuration

wicked is designed to be extensible with shell scripts. These extensions can be defined in the `config.xml` file.

Currently, several classes of extensions are supported:

- link configuration: these are scripts responsible for setting up a device's link layer according to the configuration provided by the client, and for tearing it down again.
- address configuration: these are scripts responsible for managing a device's address configuration. Usually address configuration and DHCP are managed by wicked itself, but can be implemented by means of extensions.
- firewall extension: these scripts can apply firewall rules.

Typically, extensions have a start and a stop command, an optional “pid file”, and a set of environment variables that get passed to the script.

To illustrate how this is supposed to work, look at a firewall extension defined in etc/server.xml:

```
<dbus-service interface="org.opensuse.Network.Firewall">
  <action name="firewallUp"    command="/etc/wicked/extensions/firewall up"/>
  <action name="firewallDown"  command="/etc/wicked/extensions/firewall down"/>

  <!-- default environment for all calls to this extension script -->
  <putenv name="WICKED_OBJECT_PATH" value="$object-path"/>
  <putenv name="WICKED_INTERFACE_NAME" value="$property:name"/>
  <putenv name="WICKED_INTERFACE_INDEX" value="$property:index"/>
</dbus-service>
```

The extension is attached to the <dbus-service> tag and defines commands to execute for the actions of this interface. Further, the declaration can define and initialize environment variables passed to the actions.

15.5.1.8 Wicked Extensions: Configuration Files

You can extend the handling of configuration files with scripts as well. For example, DNS updates from leases are ultimately handled by the extensions/resolver script, with behavior configured in server.xml:

```
<system-updater name="resolver">
  <action name="backup"    command="/etc/wicked/extensions/resolver backup"/>
  <action name="restore"   command="/etc/wicked/extensions/resolver restore"/>
  <action name="install"   command="/etc/wicked/extensions/resolver install"/>
```



```
<action name="remove" command="/etc/wicked/extensions/resolver remove"/>
</system-updater>
```

When an update arrives in wickedd, the system updater routines parse the lease and call the appropriate commands (backup, install, etc.) in the resolver script. This in turn configures the DNS settings using /sbin/netconfig, or by manually writing /etc/resolv.conf as a fallback.

15.5.2 Configuration Files

This section provides an overview of the network configuration files and explains their purpose and the format used.

15.5.2.1 /etc/wicked/common.xml

The /etc/wicked/common.xml file contains common definitions that should be used by all applications. It is sourced/included by the other configuration files in this directory. Even though you can use this file to, for example, enable debugging across all wicked components, we recommend to use the file /etc/wicked/local.xml for this purpose. After applying maintenance updates you might lose your changes as the /etc/wicked/common.xml might be overwritten. The /etc/wicked/common.xml file includes the /etc/wicked/local.xml in the default installation, thus you typically do not need to modify the /etc/wicked/common.xml.

In case you want to disable nanny by setting the <use-nanny> to false, restart the wickedd.service and then run the following command to apply all configurations and policies:

```
wicked ifup all
```



Note: Configuration Files

The wickedd, wicked, or nanny programs try to read /etc/wicked/common.xml if their own configuration files do not exist.

15.5.2.2 `/etc/wicked/server.xml`

The file `/etc/wicked/server.xml` is read by the `wickedd` server process at start-up. The file stores extensions to the `/etc/wicked/common.xml`. On top of that this file configures handling of a resolver and receiving information from `addrconf` supplicants, for example DHCP.

We recommend to add changes required to this file into a separate file `/etc/wicked/server-local.xml`, that gets included by `/etc/wicked/server.xml`. By using a separate file you avoid overwriting of your changes during maintenance updates.

15.5.2.3 `/etc/wicked/client.xml`

The `/etc/wicked/client.xml` is used by the `wicked` command. The file specifies the location of a script used when discovering devices managed by `ibft` and also configures locations of network interface configurations.

We recommend to add changes required to this file into a separate file `/etc/wicked/client-local.xml`, that gets included by `/etc/wicked/server.xml`. By using a separate file you avoid overwriting of your changes during maintenance updates.

15.5.2.4 `/etc/wicked/nanny.xml`

The `/etc/wicked/nanny.xml` configures types of link layers. We recommend to add specific configuration into a separate file: `/etc/wicked/nanny-local.xml` to avoid losing the changes during maintenance updates.

15.5.2.5 `/etc/sysconfig/network/ifcfg-*`

These files contain the traditional configurations for network interfaces. In SUSE Linux Enterprise 11, this was the only supported format besides `ibft` firmware.



Note: **wicked** and the `ifcfg-*` Files


wicked reads these files if you specify the `compat:` prefix. According to the SUSE Linux Enterprise Server 12 default configuration in `/etc/wicked/client.xml`, **wicked** tries these files before the XML configuration files in `/etc/wicked/ifconfig`.

The `--ifconfig` switch is provided mostly for testing only. If specified, default configuration sources defined in `/etc/wicked/ifconfig` are not applied.

The `ifcfg-*` files include information such as the start mode and the IP address. Possible parameters are described in the manual page of `ifup`. Additionally, most variables from the `dhcp` and `wireless` files can be used in the `ifcfg-*` files if a general setting should be used for only one interface. However, most of the `/etc/sysconfig/network/config` variables are global and cannot be overridden in `ifcfg`-files. For example, `NETCONFIG_*` variables are global. For configuring `macvlan` and `macvtap` interfaces, see the `ifcfg-macvlan` and `ifcfg-macvtap` man pages. For example, for a `macvlan` interface provide a `ifcfg-macvlan0` with settings as follows:

```
STARTMODE='auto'
MACVLAN_DEVICE='eth0'
#MACVLAN_MODE='vepa'
#LLADDR=02:03:04:05:06:aa
```

For `ifcfg.template`, see [Section 15.5.2.6, “/etc/sysconfig/network/config, /etc/sysconfig/network/dhcp, and /etc/sysconfig/network/wireless”](#).

System z IBM z Systems does not support USB. The names of the interface files and network aliases contain z Systems-specific elements like `qeth`. 

15.5.2.6 `/etc/sysconfig/network/config, /etc/sysconfig/network/dhcp, and /etc/sysconfig/network/wireless`

The file `config` contains general settings for the behavior of `ifup`, `ifdown` and `ifstatus`. `dhcp` contains settings for DHCP and `wireless` for wireless LAN cards. The variables in all three configuration files are commented. Some variables from `/etc/sysconfig/network/config` can also be used in `ifcfg-*` files, where they are given a higher priority. The `/etc/sysconfig/network/ifcfg.template` file lists variables that can be specified in a per interface scope. However, most of the `/etc/sysconfig/network/config` variables are global and cannot be overridden in `ifcfg`-files. For example, `NETWORKMANAGER` or `NETCONFIG_*` variables are global.



Note: Using DHCPv6

In SUSE Linux Enterprise 11, DHCPv6 used to work even on networks where IPv6 Router Advertisements (RAs) were not configured properly. Starting with SUSE Linux Enterprise 12, DHCPv6 will correctly require that at least one of the routers on the network sends out RAs that indicate that this network is managed by DHCPv6.

For those networks where the router cannot be configured correctly, there is an `ifcfg` option that allows the user to override this behavior by specifying `DHCLIENT6_MODE='managed'` in the `ifcfg` file. You can also activate this workaround with a boot parameter in the installation system:

```
ifcfg=eth0=dhcp6,DHCLIENT6_MODE=managed
```

15.5.2.7 `/etc/sysconfig/network/routes` and `/etc/sysconfig/network/ifroute-*`

The static routing of TCP/IP packets is determined by the `/etc/sysconfig/network/routes` and `/etc/sysconfig/network/ifroute-*` files. All the static routes required by the various system tasks can be specified in `/etc/sysconfig/network/routes`: routes to a host, routes to a host via a gateway and routes to a network. For each interface that needs individual routing, define an additional configuration file: `/etc/sysconfig/network/ifroute-*`. Replace the wild card (`*`) with the name of the interface. The entries in the routing configuration files look like this:

#	Destination	Gateway	Netmask	Interface	Options
---	-------------	---------	---------	-----------	---------

The route's destination is in the first column. This column may contain the IP address of a network or host or, in the case of *reachable* name servers, the fully qualified network or host name. The network should be written in CIDR notation (address with the associated routing prefix-length) such as `10.10.0.0/16` for IPv4 or `fc00::/7` for IPv6 routes. The keyword `default` indicates that the route is the default gateway in the same address family as the gateway. For devices without a gateway use explicit `0.0.0.0/0` or `::/0` destinations.

The second column contains the default gateway or a gateway through which a host or network can be accessed.

The third column is deprecated; it used to contain the IPv4 netmask of the destination. For IPv6 routes, the default route, or when using a prefix-length (CIDR notation) in the first column, enter a dash (-) here.

The fourth column contains the name of the interface. If you leave it empty using a dash (-), it can cause unintended behavior in `/etc/sysconfig/network/routes`. For more information, see the `routes` man page.

An (optional) fifth column can be used to specify special options. For details, see the `routes` man page.

EXAMPLE 15.5: COMMON NETWORK INTERFACES AND SOME STATIC ROUTES

# --- IPv4 routes in CIDR prefix notation:				
# Destination	[Gateway]	-		Interface
127.0.0.0/8	-	-		lo
204.127.235.0/24	-	-		eth0
default	204.127.235.41	-		eth0
207.68.156.51/32	207.68.145.45	-		eth1
192.168.0.0/16	207.68.156.51	-		eth1
# --- IPv4 routes in deprecated netmask notation"				
# Destination	[Dummy/Gateway]	Netmask		Interface
#				
127.0.0.0	0.0.0.0	255.255.255.0		lo
204.127.235.0	0.0.0.0	255.255.255.0		eth0
default	204.127.235.41	0.0.0.0		eth0
207.68.156.51	207.68.145.45	255.255.255.255		eth1
192.168.0.0	207.68.156.51	255.255.0.0		eth1
# --- IPv6 routes are always using CIDR notation:				
# Destination	[Gateway]	-		Interface
2001:DB8:100::/64	-	-		eth0
2001:DB8:100::/32	fe80::216:3eff:fe6d:c042	-		eth0

15.5.2.8 `/etc/resolv.conf`

The domain to which the host belongs is specified in `/etc/resolv.conf` (keyword `search`). Up to six domains with a total of 256 characters can be specified with the `search` option. When resolving a name that is not fully qualified, an attempt is made to generate one by attaching the

individual search entries. Up to 3 name servers can be specified with the nameserver option, each on a line of its own. Comments are preceded by hash mark or semicolon signs (# or ;). As an example, see *Example 15.6, "/etc/resolv.conf"*.

However, the /etc/resolv.conf should not be edited by hand. Instead, it is generated by the **netconfig** script. To define static DNS configuration without using YaST, edit the appropriate variables manually in the /etc/sysconfig/network/config file:

NETCONFIG_DNS_STATIC_SEARCHLIST

list of DNS domain names used for host name lookup

NETCONFIG_DNS_STATIC_SERVERS

list of name server IP addresses to use for host name lookup

NETCONFIG_DNS_FORWARDER

the name of the DNS forwarder that needs to be configured, for example bind or resolver

NETCONFIG_DNS_RESOLVER_OPTIONS

arbitrary options that will be written to /etc/resolv.conf, for example:

```
debug attempts:1 timeout:10
```

For more information, see the resolv.conf man page.

NETCONFIG_DNS_RESOLVER_SORTLIST

list of up to 10 items, for example:

```
130.155.160.0/255.255.240.0 130.155.0.0
```

For more information, see the resolv.conf man page.

To disable DNS configuration using netconfig, set NETCONFIG_DNS_POLICY=''. For more information about **netconfig**, see the netconfig(8) man page (**man 8 netconfig**).

EXAMPLE 15.6: /etc/resolv.conf

```
# Our domain
search example.com
#
# We use dns.example.com (192.168.1.116) as nameserver
```

15.5.2.9 `/sbin/netconfig`

netconfig is a modular tool to manage additional network configuration settings. It merges statically defined settings with settings provided by autoconfiguration mechanisms as DHCP or PPP according to a predefined policy. The required changes are applied to the system by calling the netconfig modules that are responsible for modifying a configuration file and restarting a service or a similar action.

netconfig recognizes three main actions. The **netconfig modify** and **netconfig remove** commands are used by daemons such as DHCP or PPP to provide or remove settings to netconfig. Only the **netconfig update** command is available for the user:

modify

The **netconfig modify** command modifies the current interface and service specific dynamic settings and updates the network configuration. Netconfig reads settings from standard input or from a file specified with the `--lease-file filename` option and internally stores them until a system reboot (or the next modify or remove action). Already existing settings for the same interface and service combination are overwritten. The interface is specified by the `-i interface_name` parameter. The service is specified by the `-s service_name` parameter.

remove

The **netconfig remove** command removes the dynamic settings provided by a modificatory action for the specified interface and service combination and updates the network configuration. The interface is specified by the `-i interface_name` parameter. The service is specified by the `-s service_name` parameter.

update

The **netconfig update** command updates the network configuration using current settings. This is useful when the policy or the static configuration has changed. Use the `-m module_type` parameter, if you want to update a specified service only (`dns`, `nis`, or `ntp`).

The netconfig policy and the static configuration settings are defined either manually or using YaST in the `/etc/sysconfig/network/config` file. The dynamic configuration settings provided by autoconfiguration tools such as DHCP or PPP are delivered directly by these tools with

the `netconfig modify` and `netconfig remove` actions. When NetworkManager is enabled, netconfig (in policy mode `auto`) uses only NetworkManager settings, ignoring settings from any other interfaces configured using the traditional ifup method. If NetworkManager does not provide any setting, static settings are used as a fallback. A mixed usage of NetworkManager and the `wicked` method is not supported.

For more information about `netconfig`, see `man 8 netconfig`.

15.5.2.10 `/etc/hosts`

In this file, shown in *Example 15.7, “/etc/hosts”*, IP addresses are assigned to host names. If no name server is implemented, all hosts to which an IP connection will be set up must be listed here. For each host, enter a line consisting of the IP address, the fully qualified host name, and the host name into the file. The IP address must be at the beginning of the line and the entries separated by blanks and tabs. Comments are always preceded by the `#` sign.

EXAMPLE 15.7: `/etc/hosts`

```
127.0.0.1 localhost
192.168.2.100 jupiter.example.com jupiter
192.168.2.101 venus.example.com venus
```

15.5.2.11 `/etc/networks`

Here, network names are converted to network addresses. The format is similar to that of the `hosts` file, except the network names precede the addresses. See *Example 15.8, “/etc/networks”*.

EXAMPLE 15.8: `/etc/networks`

```
loopback    127.0.0.0
localnet    192.168.0.0
```

15.5.2.12 `/etc/host.conf`

Name resolution—the translation of host and network names via the *resolver* library—is controlled by this file. This file is only used for programs linked to libc4 or libc5. For current glibc programs, refer to the settings in `/etc/nsswitch.conf`. Each parameter must always be en-

tered on a separate line. Comments are preceded by a `#` sign. *Table 15.2, "Parameters for `/etc/host.conf`"* shows the parameters available. A sample `/etc/host.conf` is shown in *Example 15.9, "`/etc/host.conf`".*

TABLE 15.2: PARAMETERS FOR `/ETC/HOST.CONF`

<code>order hosts, bind</code>	Specifies in which order the services are accessed for the name resolution. Available arguments are (separated by blank spaces or commas):
	<i>hosts</i> : searches the <code>/etc/hosts</code> file
	<i>bind</i> : accesses a name server
	<i>nis</i> : uses NIS
<code>multi on/off</code>	Defines if a host entered in <code>/etc/hosts</code> can have multiple IP addresses.
<code>nospoof on spoofalert on/off</code>	These parameters influence the name server <i>spoofing</i> but do not exert any influence on the network configuration.
<code>trim domainname</code>	The specified domain name is separated from the host name after host name resolution (as long as the host name includes the domain name). This option is useful only if names from the local domain are in the <code>/etc/hosts</code> file, but should still be recognized with the attached domain names.

EXAMPLE 15.9: `/etc/host.conf`

```
# We have named running
order hosts bind
# Allow multiple address
multi on
```

15.5.2.13 `/etc/nsswitch.conf`

The introduction of the GNU C Library 2.0 was accompanied by the introduction of the *Name Service Switch* (NSS). Refer to the `nsswitch.conf(5)` man page and *The GNU C Library Reference Manual* for details.

The order for queries is defined in the file `/etc/nsswitch.conf`. A sample `nsswitch.conf` is shown in *Example 15.10, “`/etc/nsswitch.conf`”*. Comments are preceded by `#` signs. In this example, the entry under the `hosts` database means that a request is sent to `/etc/hosts (files)` via DNS (see *Chapter 23, The Domain Name System*).

EXAMPLE 15.10: `/etc/nsswitch.conf`

```
passwd:      compat
group:       compat

hosts:       files dns
networks:    files dns

services:    db files
protocols:   db files
rpc:         files
ethers:      files
netmasks:   files
netgroup:    files nis
publickey:   files

bootparams:  files
automount:   files nis
aliases:     files nis
shadow:      compat
```

The “databases” available over NSS are listed in *Table 15.3, “Databases Available via `/etc/nsswitch.conf`”*.

The configuration options for NSS databases are listed in *Table 15.4, “Configuration Options for NSS “Databases””*.

TABLE 15.3: DATABASES AVAILABLE VIA `/ETC/NSSWITCH.CONF`

<u>aliases</u>	Mail aliases implemented by <code>sendmail</code> ; see <code>man 5 aliases</code> .
<u>ethers</u>	Ethernet addresses.

<u>netmasks</u>	List of networks and their subnet masks. Only needed, if you use subnetting.
<u>group</u>	User groups used by <u>getgrent</u> . See also the man page for <u>group</u> .
<u>hosts</u>	Host names and IP addresses, used by <u>gethostbyname</u> and similar functions.
<u>netgroup</u>	Valid host and user lists in the network for controlling access permissions; see the <u>netgroup(5)</u> man page.
<u>networks</u>	Network names and addresses, used by <u>getnetent</u> .
<u>publickey</u>	Public and secret keys for Secure_RPC used by NFS and NIS+.
<u>passwd</u>	User passwords, used by <u>getpwent</u> ; see the <u>passwd(5)</u> man page.
<u>protocols</u>	Network protocols, used by <u>getprotoent</u> ; see the <u>protocols(5)</u> man page.
<u>rpc</u>	Remote procedure call names and addresses, used by <u>getrpcbyname</u> and similar functions.
<u>services</u>	Network services, used by <u>getservent</u> .
<u>shadow</u>	Shadow passwords of users, used by <u>getspnam</u> ; see the <u>shadow(5)</u> man page.

TABLE 15.4: CONFIGURATION OPTIONS FOR NSS “DATABASES”

<u>files</u>	directly access files, for example, <u>/etc/aliases</u>
<u>db</u>	access via a database

<u>nis</u> , <u>nisplus</u>	NIS, see also <i>Book “Security Guide”, Chapter 3 “Using NIS”</i>
<u>dns</u>	can only be used as an extension for <u>hosts</u> and <u>networks</u>
<u>compat</u>	can only be used as an extension for <u>passwd</u> , <u>shadow</u> and <u>group</u>

15.5.2.14 `/etc/nscd.conf`

This file is used to configure nscd (name service cache daemon). See the nscd(8) and nscd.conf(5) man pages. By default, the system entries of passwd, groups and hosts are cached by nscd. This is important for the performance of directory services, like NIS and LDAP, because otherwise the network connection needs to be used for every access to names, groups or hosts.

If the caching for passwd is activated, it usually takes about fifteen seconds until a newly added local user is recognized. Reduce this waiting time by restarting nscd with:

```
systemctl restart nscd
```

15.5.2.15 `/etc/HOSTNAME`

/etc/HOSTNAME contains the fully qualified host name (FQHN). The fully qualified host name is the host name with the domain name attached. This file must contain only one line (in which the host name is set). It is read while the machine is booting.

15.5.3 Testing the Configuration

Before you write your configuration to the configuration files, you can test it. To set up a test configuration, use the ip command. To test the connection, use the ping command.

The command ip changes the network configuration directly without saving it in the configuration file. Unless you enter your configuration in the correct configuration files, the changed network configuration is lost on reboot.



Note: **ifconfig** and **route** Are Obsolete

The **ifconfig** and **route** tools are obsolete. Use **ip** instead. **ifconfig**, for example, limits interface names to 9 characters.

15.5.3.1 Configuring a Network Interface with **ip**

ip is a tool to show and configure network devices, routing, policy routing, and tunnels.

ip is a very complex tool. Its common syntax is **ip options object command**. You can work with the following objects:

link

This object represents a network device.

address

This object represents the IP address of device.

neighbor

This object represents an ARP or NDISC cache entry.

route

This object represents the routing table entry.

rule

This object represents a rule in the routing policy database.

maddress

This object represents a multicast address.

mroute

This object represents a multicast routing cache entry.

tunnel

This object represents a tunnel over IP.

If no command is given, the default command is used (usually **list**).

Change the state of a device with the command **ip link set device_name _**. For example, to deactivate device eth0, enter **ip link set eth0 down**. To activate it again, use **ip link set eth0 up**.

After activating a device, you can configure it. To set the IP address, use `ip addr add ip_address + dev device_name`. For example, to set the address of the interface `eth0` to `192.168.12.154/30` with standard broadcast (option `brd`), enter `ip addr add 192.168.12.154/30 brd + dev eth0`.

To have a working connection, you must also configure the default gateway. To set a gateway for your system, enter `ip route add gateway_ip_address`. To translate one IP address to another, use `nat: ip route add nat ip_address via other_ip_address`.

To display all devices, use `ip link ls`. To display the running interfaces only, use `ip link ls up`. To print interface statistics for a device, enter `ip -s link ls device_name`. To view addresses of your devices, enter `ip addr`. In the output of the `ip addr`, also find information about MAC addresses of your devices. To show all routes, use `ip route show`.

For more information about using `ip`, enter `ip help` or see the `ip(8)` man page. The `help` option is also available for all `ip` subcommands. If, for example, you need help for `ip addr`, enter `ip addr help`. Find the `ip` manual in `/usr/share/doc/packages/iproute2/ip-cref.pdf`.

15.5.3.2 Testing a Connection with ping

The `ping` command is the standard tool for testing whether a TCP/IP connection works. It uses the ICMP protocol to send a small data packet, `ECHO_REQUEST` datagram, to the destination host, requesting an immediate reply. If this works, `ping` displays a message to that effect. This indicates that the network link is functioning.

`ping` does more than only test the function of the connection between two computers: it also provides some basic information about the quality of the connection. In *Example 15.11, "Output of the Command ping"*, you can see an example of the `ping` output. The second-to-last line contains information about the number of transmitted packets, packet loss, and total time of `ping` running.

As the destination, you can use a host name or IP address, for example, `ping example.com` or `ping 192.168.3.100`. The program sends packets until you press `Ctrl-C`.

If you only need to check the functionality of the connection, you can limit the number of the packets with the `-c` option. For example to limit ping to three packets, enter `ping -c 3 example.com`.

EXAMPLE 15.11: OUTPUT OF THE COMMAND PING

```
ping -c 3 example.com
```

```
PING example.com (192.168.3.100) 56(84) bytes of data.  
64 bytes from example.com (192.168.3.100): icmp_seq=1 ttl=49 time=188 ms  
64 bytes from example.com (192.168.3.100): icmp_seq=2 ttl=49 time=184 ms  
64 bytes from example.com (192.168.3.100): icmp_seq=3 ttl=49 time=183 ms  
--- example.com ping statistics ---  
3 packets transmitted, 3 received, 0% packet loss, time 2007ms  
rtt min/avg/max/mdev = 183.417/185.447/188.259/2.052 ms
```

The default interval between two packets is one second. To change the interval, ping provides the option `-i`. For example, to increase the ping interval to ten seconds, enter `ping -i 10 example.com`.

In a system with multiple network devices, it is sometimes useful to send the ping through a specific interface address. To do so, use the `-I` option with the name of the selected device, for example, `ping -I wlan1 example.com`.

For more options and information about using ping, enter `ping -h` or see the `ping (8)` man page.



Tip: Pinging IPv6 Addresses

For IPv6 addresses use the `ping6` command. Note, to ping link-local addresses, you must specify the interface with `-I`. The following command works, if the address is reachable via `eth1`:

```
ping6 -I eth1 fe80::117:21ff:feda:a425
```

15.5.4 Unit Files and Start-Up Scripts

Apart from the configuration files described above, there are also systemd unit files and various scripts that load the network services while the machine is booting. These are started when the system is switched to the `multi-user.target` target. Some of these unit files and scripts are described in *Some Unit Files and Start-Up Scripts for Network Programs*. For more information about `systemd`, see *Chapter 13, The systemd Daemon* and for more information about the `systemd` targets, see the man page of `systemd.special` (`man systemd.special`).

SOME UNIT FILES AND START-UP SCRIPTS FOR NETWORK PROGRAMS

network.target

network.target is the systemd target for networking, but its meaning depends on the settings provided by the system administrator.

For more information, see <http://www.freedesktop.org/wiki/Software/systemd/NetworkTarget/>.

multi-user.target

multi-user.target is the systemd target for a multiuser system with all required network services.

xinetd

Starts xinetd. xinetd can be used to make server services available on the system. For example, it can start vsftpd whenever an FTP connection is initiated.

rpcbind

Starts the rpcbind utility that converts RPC program numbers to universal addresses. It is needed for RPC services, such as an NFS server.

ypserv

Starts the NIS server.

ypbind

Starts the NIS client.

/etc/init.d/nfsserver

Starts the NFS server.

/etc/init.d/postfix

Controls the postfix process.

15.6 Basic Router Setup

A router is a networking device that delivers and receives data (network packets) to or from more than one network back and forth. You often use a router to connect your local network to the remote network (Internet) or to connect local network segments. With SUSE Linux Enterprise Server you can build a router with features such as NAT (Network Address Translation) or advanced firewalling.

The following are basic steps to turn SUSE Linux Enterprise Server into a router.

1. Enable forwarding, for example in `/etc/sysctl.d/50-router.conf`

```
net.ipv4.conf.all.forwarding = 1
net.ipv6.conf.all.forwarding = 1
```

Then provide a static IPv4 and IPv6 IP setup for the interfaces. Enabling forwarding disables several mechanisms, for example IPv6 does not accept an IPv6 RA (router advertisement) anymore, which also prevents the creation of a default route.

2. In many situations, such as when you can reach the same (internal) network via more than one interface, or when VPN is usually used (and already on “normal multi-home hosts”), you must disable the IPv4 reverse path filter (this feature does not currently exist for IPv6):

```
net.ipv4.conf.all.rp_filter = 0
```

You can also filter with firewall settings instead.

3. To accept an IPv6 RA (from the router on an external, uplink, or ISP interface) and create a default (or also a more specific) IPv6 route again, set:

```
net.ipv6.conf.${ifname}.accept_ra = 2
net.ipv6.conf.${ifname}.autoconf = 0
```

(Note: “`eth0.42`” needs to be written as `eth0/42` in a dot-separated sysfs path.)

More router behavior and forwarding dependencies are described in <https://www.kernel.org/doc/Documentation/networking/ip-sysctl.txt>.

To provide IPv6 on your internal (DMZ) interfaces, and announce yourself as an IPv6 router and “autoconf networks” to the clients, install and configure `radvd` in `/etc/radvd.conf`, for example:

```
interface eth0
{
    IgnoreIfMissing on;           # do not fail if interface missed

    AdvSendAdvert on;            # enable sending RAs
    AdvManagedFlag on;          # IPv6 addresses managed via DHCPv6
    AdvOtherConfigFlag on;       # DNS, NTP... only via DHCPv6

    AdvDefaultLifetime 3600;     # client default route lifetime of 1 hour
}
```

```

prefix 2001:db8:0:1::/64    # (/64 is default and required for autoconf)
{
    AdvAutonomous off;      # Disable address autoconf (DHCPv6 only)

    AdvValidLifetime 3600;  # prefix (autoconf addr) is valid 1 h
    AdvPreferredLifetime 1800; # prefix (autoconf addr) is preferred 1/2 h
}
}

```

Lastly configure the firewall. In `SuSEfirewall2`, you need to set `FW_ROUTE="yes"` (otherwise it will also reset forwarding `sysctl` again) and define the interfaces in the `FW_DEV_INT`, `FW_DEV_EXT` (and `FW_DEV_DMZ`) zone variables as needed, perhaps also `FW_MASQUERADE="yes"` and `FW_MASQ_DEV`.

15.7 Setting Up Bonding Devices

For some systems, there is a desire to implement network connections that comply to more than the standard data security or availability requirements of a typical Ethernet device. In these cases, several Ethernet devices can be aggregated to a single bonding device.

The configuration of the bonding device is done by means of bonding module options. The behavior is mainly affected by the mode of the bonding device. By default, this is `mode=active-backup` which means that a different slave device will become active if the active slave fails.



Tip: Bonding and Xen

Using bonding devices is only of interest for machines where you have multiple real network cards available. In most configurations, this means that you should use the bonding configuration only in Dom0. Only if you have multiple network cards assigned to a VM Guest system it may also be useful to set up the bond in a VM Guest.

To configure a bonding device, use the following procedure:

1. Run *YaST* > *System* > *Network Settings*.
2. Use *Add* and change the *Device Type* to *Bond*. Proceed with *Next*.

Network Card Setup

General Address Hardware Bond Slaves

Device Type: Bond

Configuration Name: bond0

☐ No Link and IP Setup (Bonding Slaves)

☒ Dynamic Address: DHCP DHCP both version 4 and 6

☐ Statically assigned IP Address

IP Address: Subnet Mask: 255.255.255.0 Hostname:

Additional Addresses

IPv4 Address Label	IP Address	Netmask

Add Edit Delete

Help Cancel Back Next

3. Select how to assign the IP address to the bonding device. Three methods are at your disposal:

- No IP Address
- Dynamic Address (with DHCP or Zeroconf)
- Statically assigned IP Address

Use the method that is appropriate for your environment.

4. In the *Bond Slaves* tab, select the Ethernet devices that should be included into the bond by activating the related check box.

5. Edit the *Bond Driver Options*. The modes that are available for configuration are the following:

- balance-rr
- active-backup
- balance-xor
- broadcast
- 802.3ad

802.3ad is the standardized LACP “IEEE 802.3ad Dynamic link aggregation” mode.

- balance-tlb
 - balance-alb
6. Make sure that the parameter miimon=100 is added to the *Bond Driver Options*. Without this parameter, the data integrity is not checked regularly.
 7. Click *Next* and leave YaST with *OK* to create the device.

All modes, and many more options are explained in detail in the *Linux Ethernet Bonding Driver HOWTO* found at /usr/src/linux/Documentation/networking/bonding.txt after installing the package kernel-source.

15.7.1 Hotplugging of Bonding Slaves

In specific network environments (such as High Availability), there are cases when you need to replace a bonding slave interface with another one. The reason may be a constantly failing network device. The solution is to set up hotplugging of bonding slaves.

The bond is configured as usual (according to man 5 ifcfg-bonding), for example:

```
ifcfg-bond0
    STARTMODE='auto' # or 'onboot'
    BOOTPROTO='static'
    IPADDR='192.168.0.1/24'
    BONDING_MASTER='yes'
    BONDING_SLAVE_0='eth0'
    BONDING_SLAVE_1='eth1'
    BONDING_MODULE_OPTS='mode=active-backup miimon=100'
```

The slaves are specified with STARTMODE=hotplug and BOOTPROTO=none:

```
ifcfg-eth0
    STARTMODE='hotplug'
    BOOTPROTO='none'

ifcfg-eth1
    STARTMODE='hotplug'
    BOOTPROTO='none'
```

`BOOTPROTO=none` uses the **ethtool** options (when provided), but does not set the link up on **ifup eth0**. The reason is that the slave interface is controlled by the bond master.

`STARTMODE=hotplug` causes the slave interface to join the bond automatically when it is available.

The `udev` rules in `/etc/udev/rules.d/70-persistent-net.rules` need to be changed to match the device by bus ID (`udev KERNELS` keyword equal to "SysFS BusID" as visible in **hwinfo --netcard**) instead of by MAC address to allow to replacement of defective hardware (a network card in the same slot but with a different MAC), and to avoid confusion as the bond changes the MAC address of all its slaves.

For example:

```
SUBSYSTEM=="net", ACTION=="add", DRIVERS=="*",
KERNELS=="0000:00:19.0", ATTR{dev_id}=="0x0", ATTR{type}=="1",
KERNEL=="eth*", NAME="eth0"
```

At boot time, the `systemd network.service` does not wait for the hotplug slaves, but for the bond to become ready, which requires at least one available slave. When one of the slave interfaces gets removed (unbind from NIC driver, **rmmod** of the NIC driver or true PCI hotplug remove) from the system, the kernel removes it from the bond automatically. When a new card is added to the system (replacement of the hardware in the slot), `udev` renames it using the bus-based persistent name rule to the name of the slave, and calls **ifup** for it. The **ifup** call automatically joins it into the bond.

15.8 Setting Up Team Devices for Network Teaming

The term “link aggregation” is the general term which describes combining (or aggregating) a network connection to provide a logical layer. Sometimes you find the terms “channel teaming”, “Ethernet bonding”, “port truncating”, etc. which are synonyms and refer to the same concept. This concept is widely known as “bonding” and was originally integrated into the Linux kernel (see [Section 15.7, “Setting Up Bonding Devices”](#) for the original implementation). The term *Network Teaming* is used to refer to the new implementation of this concept.

The main difference between bonding and Network Teaming is that teaming supplies a set of small kernel modules which are responsible for providing an interface for teamd instances. Everything else is handled in user space. This is different from the original bonding implementation which contains all of its functionality exclusively in the kernel.

Both implementations, bonding and Network Teaming, can be used in parallel. Network Teaming is an alternative to the existing bonding implementation. It does not replace bonding.

Network Teaming can be used for different use cases. The two most important use cases are explained later and involve:

- Load balancing between different network devices.
- Failover from one network device to another in case one of the devices should fail.

Currently, there is no YaST module to support creating a teaming device. You need to configure Network Teaming manually. The general procedure is shown below which can be applied for all your Network Teaming configurations:

PROCEDURE 15.1: GENERAL PROCEDURE

1. Make sure you have all the necessary packages installed. Install the packages `libteam-tools` , `libteamctl0` , `libteamctl0` , and `python-libteam` .
2. Create a configuration file under `/etc/sysconfig/network/`. Usually it will be `ifcfg-team0` . If you need more than one Network Teaming device, give them ascending numbers.

This configuration file contains several variables which are explained in the man pages (see `man ifcfg` and `man ifcfg-team`).

3. Remove the configuration files of the interfaces which will be used for the teaming device (usually `ifcfg-eth0` and `ifcfg-eth1`).

It is recommended to make a backup and remove both files. Wicked will re-create the configuration files with the necessary parameters for teaming.

4. Optionally, check if everything is included in Wicked's configuration file:

```
wicked show-config
```

5. Start the Network Teaming device `team0`:

```
wicked all ifup team0
```

In case you need additional debug information, use the option `--debug all` after the `all` subcommand.

6. Check the status of the Network Teaming device. This can be done by the following commands:

- Get the state of the teamd instance from Wicked:

```
wicked ifstatus --verbose team0
```

- Get the state of the entire instance:

```
teamdctl team0 state
```

- Get the systemd state of the teamd instance:

```
systemctl status teamd@team0
```

Each of them shows a slightly different view depending on your needs.

7. In case you need to change something in the `ifcfg-team0` file afterward, reload its configuration with:

```
wicked ifreload team0
```

Do *not* use `systemctl` for starting or stopping the teaming device! Instead, use the `wicked` command as shown above.

15.8.1 Use Case: Loadbalancing with Network Teaming

Loadbalancing is used to improve bandwidth. Use the following configuration file to create a Network Teaming device with loadbalancing capabilities. Proceed with *Procedure 15.1, "General Procedure"* to set up the device. Check the output with `teamdctl`.

EXAMPLE 15.12: CONFIGURATION FOR LOADBALANCING WITH NETWORK TEAMING

```
STARTMODE=auto ❶  
BOOTPROTO=static ❷  
IPADDRESS="192.168.1.1/24" ❷  
IPADDR6="fd00:deca:fbad:50::1/64" ❷
```

```

TEAM_RUNNER="loadbalance" ❸
TEAM_LB_TX_HASH="ipv4,ipv6,eth,vlan"
TEAM_LB_TX_BALANCER_NAME="basic"
TEAM_LB_TX_BALANCER_INTERVAL="100"

TEAM_PORT_DEVICE_0="eth0" ❹
TEAM_PORT_DEVICE_1="eth1" ❹

TEAM_LW_NAME="ethtool" ❺
TEAM_LW_ETHTOOL_DELAY_UP="10" ❻
TEAM_LW_ETHTOOL_DELAY_DOWN="10" ❻

```

- ❶ Controls the start of the teaming device. The value of auto means, the interface will be set up when the network service is available and will be started automatically on every reboot. In case you need to control the device yourself (and prevent it from starting automatically), set STARTMODE to manual.
- ❷ Sets a static IP address (here 192.168.1.1 for IPv4 and fd00:deca:fbad:50::1 for IPv6).
If the Network Teaming device should use a dynamic IP address, set BOOTPROTO="dhcp" and remove (or comment) the line with IPADDRESS and IPADDR6.
- ❸ Sets TEAM_RUNNER to loadbalance to activate the loadbalancing mode.
- ❹ Specifies one or more devices which should be aggregated to create the Network Teaming device.
- ❺ Defines a link watcher to monitor the state of subordinate devices. The default value ethtool checks only if the device is up and accessible. This makes this check fast enough. However, it does not check if the device can really send or receive packets.
If you need a higher confidence in the connection, use the arp_ping option. This sends pings to an arbitrary host (configured in the TEAM_LW_ARP_PING_TARGET_HOST variable). Only if the replies are received, the Network Teaming device is considered to be up.
- ❻ Defines the delay in milliseconds between the link coming up (or down) and the runner being notified.

15.8.2 Use Case: Failover with Network Teaming

Failover is used to ensure high availability of a critical Network Teaming device by involving a parallel backup network device. The backup network device is running all the time and takes over if and when the main device fails.

Use the following configuration file to create a Network Teaming device with failover capabilities. Proceed with *Procedure 15.1, "General Procedure"* to set up the device. Check the output with **teamctl**.

EXAMPLE 15.13: CONFIGURATION FOR DHCP NETWORK TEAMING DEVICE

```
STARTMODE=auto ❶
BOOTPROTO=static ❷
IPADDR="192.168.1.2/24" ❷
IPADDR6="fd00:deca:fbad:50::2/64" ❷

TEAM_RUNNER=activebackup ❸
TEAM_PORT_DEVICE_0="eth0" ❹
TEAM_PORT_DEVICE_1="eth1" ❹

TEAM_LW_NAME=ethtool ❺
TEAM_LW_ETHTOOL_DELAY_UP="10" ❻
TEAM_LW_ETHTOOL_DELAY_DOWN="10" ❻
```

- ❶ Controls the start of the teaming device. The value of auto means, the interface will be set up when the network service is available and will be started automatically on every reboot. In case you need to control the device yourself (and prevent it from starting automatically), set STARTMODE to manual.
- ❷ Sets a static IP address (here 192.168.1.2 for IPv4 and fd00:deca:fbad:50::2 for IPv6).
If the Network Teaming device should use a dynamic IP address, set BOOTPROTO="dhcp" and remove (or comment) the line with IPADDRESS and IPADDR6.
- ❸ Sets TEAM_RUNNER to activebackup to activate the failover mode.
- ❹ Specifies one or more devices which should be aggregated to create the Network Teaming device.
- ❺ Defines a link watcher to monitor the state of subordinate devices. The default value eth-tool checks only if the device is up and accessible. This makes this check fast enough. However, it does not check if the device can really send or receive packets.
If you need a higher confidence in the connection, use the arp_ping option. This sends pings to an arbitrary host (configured in the TEAM_LW_ARP_PING_TARGET_HOST variable). Only if the replies are received, the Network Teaming device is considered to be up.
- ❻ Defines the delay in milliseconds between the link coming up (or down) and the runner being notified.

15.9 Software-Defined Networking with Open vSwitch

Software-defined networking (SDN) means separating the system that controls where traffic is sent (the *control plane*) from the underlying system that forwards traffic to the selected destination (the *data plane*, also called the *forwarding plane*). This means that the functions previously fulfilled by a single, usually inflexible switch can now be separated between a switch (data plane) and its controller (control plane). In this model, the controller is programmable and can be very flexible and adapt quickly to changing network conditions.

Open vSwitch is software that implements a distributed virtual multilayer switch that is compatible with the OpenFlow protocol. OpenFlow allows a controller application to modify the configuration of a switch. OpenFlow is layered onto the TCP protocol and is implemented in a range of hardware and software. A single controller can thus drive multiple, very different switches.

15.9.1 Advantages of Open vSwitch

Software-defined networking with Open vSwitch brings several advantages with it, especially when you used together with virtual machines:

- Networking states can be identified easily.
- Networks and their live state can be moved from one host to another.
- Network dynamics are traceable and external software can be enabled to respond to them.
- You can apply and manipulate tags in network packets to identify which machine they are coming from or going to and maintain other networking context. Tagging rules can be configured and migrated.
- Open vSwitch implements the GRE protocol (*Generic Routing Encapsulation*). This allows you, for example, to connect private VM networks to each other.
- Open vSwitch can be used on its own, but is designed to integrate with networking hardware and can control hardware switches.

15.9.2 Installing Open vSwitch

1. Install Open vSwitch and supplementary packages:

```
root # zypper install openvswitch openvswitch-switch
```

If you plan to use Open vSwitch together with the KVM hypervisor, additionally install tunctl . If you plan to use Open vSwitch together with the Xen hypervisor, additionally install openvswitch-kmp-xen .

2. Enable the Open vSwitch service:

```
root # systemctl enable openvswitch
```

3. Either restart the computer or use systemctl to start the Open vSwitch service immediately:

```
root # systemctl start openvswitch
```

4. To check whether Open vSwitch was activated correctly, use:

```
root # systemctl status openvswitch
```

15.9.3 Overview of Open vSwitch Daemons and Utilities

Open vSwitch consists of several components. Among them are a kernel module and various user space components. The kernel module is used for accelerating the data path, but is not necessary for a minimal Open vSwitch installation.

15.9.3.1 Daemons

The central executables of Open vSwitch are its two daemons. When you start the openvswitch service, you are indirectly starting them.

The main Open vSwitch daemon (ovs-vswitchd) provides the implementation of a switch. The Open vSwitch database daemon (ovsdb-server) serves the database that stores the configuration and state of Open vSwitch.

15.9.3.2 Utilities

Open vSwitch also comes with several utilities that help you work with it. The following list is not exhaustive, but instead describes important commands only.

ovsdb-tool

Create, upgrade, compact, and query Open vSwitch databases. Do transactions on Open vSwitch databases.

ovs-appctl

Configure a running **ovs-vswitchd** or **ovsdb-server** daemon.

ovs-dpctl, ovs-dpctl-top

Create, modify, visualize, and delete data paths. Using this tool can interfere with **ovs-vswitchd** also performing data path management. Therefore, it is often used for diagnostics only.

ovs-dpctl-top creates a **top**-like visualization for data paths.

ovs-ofctl

Manage any switches adhering to the OpenFlow protocol. **ovs-ofctl** is not limited to interacting with Open vSwitch.

ovs-vsctl

Provides a high-level interface to the configuration database. It can be used to query and modify the database. In effect, it shows the status of **ovs-vswitchd** and can be used to configure it.

15.9.4 Creating a Bridge with Open vSwitch

The following example configuration uses the Wicked network service that is used by default on SUSE Linux Enterprise Server. To learn more about Wicked, see [Section 15.5, “Configuring a Network Connection Manually”](#).

When you have installed and started Open vSwitch, proceed as follows:

1. To configure a bridge for use by your virtual machine, create a file with content like this:

```
STARTMODE='auto' ❶  
BOOTPROTO='dhcp' ❷
```

```
OVS_BRIDGE='yes' ③  
OVS_BRIDGE_PORT_DEVICE_1='eth0' ④
```

- ① Set up the bridge automatically when the network service is started.
- ② The protocol to use for configuring the IP address.
- ③ Mark the configuration as an Open vSwitch bridge.
- ④ Choose which device/devices should be added to the bridge. To add more devices, append additional lines for each of them to the file:

```
OVS_BRIDGE_PORT_DEVICE_SUFFIX='DEVICE'
```

The SUFFIX can be any alphanumeric string. However, to avoid overwriting a previous definition, make sure the SUFFIX of each device is unique.

Save the file in the directory /etc/sysconfig/network under the name ifcfg-br0. Instead of br0, you can use any name you want. However, the file name needs to begin with ifcfg-.

To learn about further options, refer to the man pages of ifcfg (**man 5 ifcfg**) and ifcfg-ovs-bridge (**man 5 ifcfg-ovs-bridge**).

2. Now start the bridge:

```
root # wicked ifup br0
```

When Wicked is done, it should output the name of the bridge and next to it the state up.

15.9.5 Using Open vSwitch Directly with KVM

After having created the bridge as described before in *Section 15.9.4, “Creating a Bridge with Open vSwitch”*, you can use Open vSwitch to manage the network access of virtual machines created with KVM/QEMU.

1. To be able to best use the capabilities of Wicked, make some further changes to the bridge configured before. Open the previously created /etc/sysconfig/network/ifcfg-br0 and append a line for another port device:

```
OVS_BRIDGE_PORT_DEVICE_2='tap0'
```

Additionally, set `BOOTPROTO` to `none`. The file should now look like this:

```
STARTMODE='auto'
BOOTPROTO='none'
OVS_BRIDGE='yes'
OVS_BRIDGE_PORT_DEVICE_1='eth0'
OVS_BRIDGE_PORT_DEVICE_2='tap0'
```

The new port device `tap0` will be configured in the next step.

2. Now add a configuration file for the `tap0` device:

```
STARTMODE='auto'
BOOTPROTO='none'
TUNNEL='tap'
```

Save the file in the directory `/etc/sysconfig/network` under the name `ifcfg-tap0`.



Tip: Allowing Other Users to Access the Tap Device

To be able to use this tap device from a virtual machine started as a user who is not `root`, append:

```
TUNNEL_SET_OWNER=USER_NAME
```

To allow access for an entire group, append:

```
TUNNEL_SET_GROUP=GROUP_NAME
```

3. Finally, open the configuration for the device defined as the first `OVS_BRIDGE_PORT_DEVICE`. If you did not change the name, that should be `eth0`. Therefore, open `/etc/sysconfig/network/ifcfg-eth0` and make sure that the following options are set:

```
STARTMODE='auto'
BOOTPROTO='none'
```

If the file does not exist yet, create it.

- Restart the bridge interface using Wicked:

```
root # wicked ifreload br0
```

This will also trigger a reload of the newly defined bridge port devices.

- To start a virtual machine, use, for example:

```
root # qemu-kvm \  
-drive file=/PATH/TO/DISK-IMAGE ❶ \  
-m 512 -net nic,vlan=0,macaddr=00:11:22:EE:EE:EE \  
-net tap,ifname=tap0,script=no,downscript=no ❷
```

- The path to the QEMU disk image you want to start.
- Use the tap device (`tap0`) created before.

For further information on the usage of KVM/QEMU, see *Book “Virtualization Guide”*.

15.9.6 Using Open vSwitch with libvirt

After having created the bridge as described before in *Section 15.9.4, “Creating a Bridge with Open vSwitch”*, you can add the bridge to an existing virtual machine managed with `libvirt`. Since `libvirt` has some support for Open vSwitch bridges already, you can use the bridge created in *Section 15.9.4, “Creating a Bridge with Open vSwitch”* without further changes to the networking configuration.

- Open the domain XML file for the intended virtual machine:

```
root # virsh edit VM_NAME
```

Replace `VM_NAME` with the name of the desired virtual machine. This will open your default text editor.

- Find the networking section of the document by looking for a section starting with `<interface type=“...”>` and ending in `</interface>`.

Replace the existing section with a networking section that looks somewhat like this:

```
<interface type='bridge'>  
  <source bridge='br0' />  
  <virtualport type='openvswitch' />
```

```
</interface>
```



Important: Compatibility of **virsh iface-*** and Virtual Machine Manager with Open vSwitch

At the moment, the Open vSwitch compatibility of `libvirt` is not exposed through the `virsh iface-*` tools and Virtual Machine Manager. If you use any of these tools, your configuration can break.


3. You can now start or restart the virtual machine as usual.

For further information on the usage of `libvirt`, see Book “Virtualization Guide”.

15.9.7 For More Information

<http://openvswitch.org/support/> 

The documentation section of the Open vSwitch project Web site

<https://www.opennetworking.org/images/stories/downloads/sdn-resources/white-papers/wp-sdn-newnorm.pdf> 

Whitepaper by the Open Networking Foundation about software-defined networking and the OpenFlow protocol

16 Printer Operation

SUSE® Linux Enterprise Server supports printing with many types of printers, including remote network printers. Printers can be configured manually or with YaST. For configuration instructions, refer to *Book “Deployment Guide”, Chapter 10 “Setting Up Hardware Components with YaST”, Section 10.3 “Setting Up a Printer”*. Both graphical and command line utilities are available for starting and managing print jobs. If your printer does not work as expected, refer to *Section 16.8, “Troubleshooting”*.

CUPS (Common Unix Printing System) is the standard print system in SUSE Linux Enterprise Server.

Printers can be distinguished by interface, such as USB or network, and printer language. When buying a printer, make sure that the printer has an interface that is supported (USB, Ethernet, or Wi-Fi) and a suitable printer language. Printers can be categorized on the basis of the following three classes of printer languages:

PostScript Printers

PostScript is the printer language in which most print jobs in Linux and Unix are generated and processed by the internal print system. If PostScript documents can be processed directly by the printer and do not need to be converted in additional stages in the print system, the number of potential error sources is reduced.

Currently PostScript is being replaced by PDF as the standard print job format. PostScript + PDF printers that can directly print PDF (in addition to PostScript) already exist. For traditional PostScript printers PDF needs to be converted to PostScript in the printing workflow.

Standard Printers (Languages Like PCL and ESC/P)

In the case of known printer languages, the print system can convert PostScript jobs to the respective printer language with Ghostscript. This processing stage is called interpreting. The best-known languages are PCL (which is mostly used by HP printers and their clones) and ESC/P (which is used by Epson printers). These printer languages are usually supported by Linux and produce an adequate print result. Linux may not be able to address some special printer functions. Except for HP and Epson there are currently no printer manufacturers who develop Linux drivers and make them available to Linux distributors under an open source license.

Proprietary Printers (Also Called GDI Printers)

These printers do not support any of the common printer languages. They use their own undocumented printer languages, which are subject to change when a new edition of a model is released. Usually only Windows drivers are available for these printers. See [Section 16.8.1, “Printers without Standard Printer Language Support”](#) for more information.

Before you buy a new printer, refer to the following sources to check how well the printer you intend to buy is supported:

<http://www.linuxfoundation.org/OpenPrinting/> 

The OpenPrinting home page with the printer database. The database shows the latest Linux support status. However, a Linux distribution can only integrate the drivers available at production time. Accordingly, a printer currently rated as “perfectly supported” may not have had this status when the latest SUSE Linux Enterprise Server version was released. Thus, the databases may not necessarily indicate the correct status, but only provide an approximation.

<http://pages.cs.wisc.edu/~ghost/> 

The Ghostscript Web page.

</usr/share/doc/packages/ghostscript/catalog.devices>

List of built-in Ghostscript drivers.

16.1 The CUPS Workflow


The user creates a print job. The print job consists of the data to print plus information for the spooler, such as the name of the printer or the name of the print queue, and optionally, information for the filter, such as printer-specific options.


At least one dedicated print queue exists for every printer. The spooler holds the print job in the queue until the desired printer is ready to receive data. When the printer is ready, the spooler sends the data through the filter and back-end to the printer.

The filter converts the data generated by the application that is printing (usually PostScript or PDF, but also ASCII, JPEG, etc.) into printer-specific data (PostScript, PCL, ESC/P, etc.). The features of the printer are described in the PPD files. A PPD file contains printer-specific options with the parameters needed to enable them on the printer. The filter system makes sure that options selected by the user are enabled.

If you use a PostScript printer, the filter system converts the data into printer-specific PostScript. This does not require a printer driver. If you use a non-PostScript printer, the filter system converts the data into printer-specific data. This requires a printer driver suitable for your printer. The back-end receives the printer-specific data from the filter then passes it to the printer.

16.2 Methods and Protocols for Connecting Printers

There are various possibilities for connecting a printer to the system. The configuration of CUPS does not distinguish between a local printer and a printer connected to the system over the network. For more information about the printer connection, read the article *CUPS in a Nutshell* at http://en.opensuse.org/SDB:CUPS_in_a_Nutshell .

System z Printers and similar devices provided by the z/VM that connect locally with the IBM z Systems mainframes are not supported by CUPS. On these platforms, printing is only possible over the network. The cabling for network printers must be installed according to the instructions of the printer manufacturer. .



Warning: Changing Cable Connections in a Running System

When connecting the printer to the machine, do not forget that only USB devices can be plugged in or unplugged during operation. To avoid damaging your system or printer, shut down the system before changing any connections that are not USB.

16.3 Installing the Software

PPD (PostScript printer description) is the computer language that describes the properties, like resolution, and options, such as the availability of a duplex unit. These descriptions are required for using various printer options in CUPS. Without a PPD file, the print data would be forwarded to the printer in a “raw” state, which is usually not desired.

To configure a PostScript printer, the best approach is to get a suitable PPD file. Many PPD files are available in the packages `manufacturer-PPDs` and `OpenPrintingPPDs-postscript`. See [Section 16.7.3, “PPD Files in Various Packages”](#) and [Section 16.8.2, “No Suitable PPD File Available for a PostScript Printer”](#).

New PPD files can be stored in the directory `/usr/share/cups/model/` or added to the print system with YaST as described in *Book “Deployment Guide”, Chapter 10 “Setting Up Hardware Components with YaST”, Section 10.3.1.1 “Adding Drivers with YaST”*. Subsequently, the PPD file can be selected during the printer setup.

Be careful if a printer manufacturer wants you to install entire software packages. First, this kind of installation may result in the loss of the support provided by SUSE Linux Enterprise Server and second, print commands may work differently and the system may no longer be able to address devices of other manufacturers. For this reason, the installation of manufacturer software is not recommended.

16.4 Network Printers

A network printer can support various protocols, some even concurrently. Although most of the supported protocols are standardized, some manufacturers modify the standard. Manufacturers then provide drivers for only a few operating systems. Unfortunately, Linux drivers are rarely provided. The current situation is such that you cannot act on the assumption that every protocol works smoothly in Linux. Therefore, you may need to experiment with various options to achieve a functional configuration.

CUPS supports the socket, LPD, IPP and smb protocols.

socket

Socket refers to a connection in which the plain print data is sent directly to a TCP socket. Some socket port numbers that are commonly used are 9100 or 35. The device URI (uniform resource identifier) syntax is: `socket://IP.of.the.printer:port`, for example: `socket://192.168.2.202:9100/`.

LPD (Line Printer Daemon)

The LPD protocol is described in RFC 1179. Under this protocol, some job-related data, such as the ID of the print queue, is sent before the actual print data is sent. Therefore, a print queue must be specified when configuring the LPD protocol. The implementations of diverse printer manufacturers are flexible enough to accept any name as the print queue. If necessary, the printer manual should indicate what name to use. LPT, LPT1, LP1 or similar names are often used. The port number for an LPD service is 515. An example device URI is `lpd://192.168.2.202/LPT1`.

IPP (Internet Printing Protocol)

IPP is a relatively new protocol (1999) based on the HTTP protocol. With IPP, more job-related data is transmitted than with the other protocols. CUPS uses IPP for internal data transmission. The name of the print queue is necessary to configure IPP correctly. The port number for IPP is 631. Example device URIs are ipp://192.168.2.202/ps and ipp://192.168.2.202/printers/ps.

SMB (Windows Share)

CUPS also supports printing on printers connected to Windows shares. The protocol used for this purpose is SMB. SMB uses the port numbers 137, 138 and 139. Example device URIs are smb://user:password@workgroup/smb.example.com/printer, smb://user:password@smb.example.com/printer, and smb://smb.example.com/printer.

The protocol supported by the printer must be determined before configuration. If the manufacturer does not provide the needed information, the command nmap (which comes with the nmap package) can be used to ascertain the protocol. nmap checks a host for open ports. For example:

```
nmap -p 35,137-139,515,631,9100-10000 printerIP
```

16.5 Configuring CUPS with Command Line Tools

CUPS can be configured with command line tools like lpinfo, lpadmin and lpoptions. You need a device URI consisting of a back-end, such as USB, and parameters. To determine valid device URIs on your system use the command lpinfo -v | grep "://":

```
# lpinfo -v | grep "://"
direct usb://ACME/FunPrinter%20XL
network socket://192.168.2.253
```

With lpadmin the CUPS server administrator can add, remove or manage print queues. To add a print queue, use the following syntax:

```
lpadmin -p queue -v device-URI -P PPD-file -E
```

Then the device (-v) is available as queue (-p), using the specified PPD file (-P). This means that you must know the PPD file and the device URI to configure the printer manually.

Do not use `-E` as the first option. For all CUPS commands, `-E` as the first argument sets use of an encrypted connection. To enable the printer, `-E` must be used as shown in the following example:

```
lpadmin -p ps -v usb://ACME/FunPrinter%20XL -P \
/usr/share/cups/model/Postscript.ppd.gz -E
```

The following example configures a network printer:

```
lpadmin -p ps -v socket://192.168.2.202:9100/ -P \
/usr/share/cups/model/Postscript-level1.ppd.gz -E
```

For more options of **lpadmin**, see the man page of `lpadmin(8)`.

During printer setup, certain options are set as default. These options can be modified for every print job (depending on the print tool used). Changing these default options with YaST is also possible. Using command line tools, set default options as follows:

1. First, list all options:

```
lpoptions -p queue -l
```

Example:

```
Resolution/Output Resolution: 150dpi *300dpi 600dpi
```

The activated default option is identified by a preceding asterisk (*).

2. Change the option with **lpadmin**:

```
lpadmin -p queue -o Resolution=600dpi
```

3. Check the new setting:

```
lpoptions -p queue -l
```

```
Resolution/Output Resolution: 150dpi 300dpi *600dpi
```

When a normal user runs **lpoptions**, the settings are written to `~/.cups/lpoptions`. However, `root` settings are written to `/etc/cups/lpoptions`.

16.6 Printing from the Command Line

To print from the command line, enter `lp -d queuename filename`, substituting the corresponding names for queuename and filename.

Some applications rely on the `lp` command for printing. In this case, enter the correct command in the application's print dialog, usually without specifying filename, for example, `lp -d queuename`.

16.7 Special Features in SUSE Linux Enterprise Server

Several CUPS features have been adapted for SUSE Linux Enterprise Server. Some of the most important changes are covered here.

16.7.1 CUPS and Firewall

After having performed a default installation of SUSE Linux Enterprise Server, SuSEFirewall2 is active and the network interfaces are configured to be in the External Zone which blocks incoming traffic. More information about the SuSEFirewall2 configuration is available in *Book “Security Guide”, Chapter 15 “Masquerading and Firewalls”, Section 15.4 “SuSEFirewall2”* and at http://en.opensuse.org/SDB:CUPS_and_SANE_Firewall_settings.

16.7.1.1 CUPS Client

Normally, a CUPS client runs on a regular workstation located in a trusted network environment behind a firewall. In this case it is recommended to configure the network interface to be in the Internal Zone, so the workstation is reachable from within the network.

16.7.1.2 CUPS Server

If the CUPS server is part of a trusted network environment protected by a firewall, the network interface should be configured to be in the Internal Zone of the firewall. It is not recommended to set up a CUPS server in an untrusted network environment unless you take care that it is protected by special firewall rules and secure settings in the CUPS configuration.

16.7.2 Browsing for Network Printers

CUPS servers regularly announce the availability and status information of shared printers over the network. Clients can access this information to display a list of available printers in printing dialogs, for example. This is called “browsing”.

CUPS servers announce their print queues over the network either via the traditional CUPS browsing protocol or via Bonjour/DND-SD. To be able to browse network print queues, the service cups-browsed needs to run on all clients that print via CUPS servers. cups-browsed is not started by default. To start it for the active session, use **sudo systemctl start cups-browsed**. To ensure it is automatically started after booting, enable it with **sudo systemctl enable cups-browsed** on all clients.

In case browsing does not work after having started cups - browsed, the CUPS server(s) probably announce the network print queues via Bonjour/DND-SD. In this case you need to additionally install the package avahi and start the associated service with **sudo systemctl start avahi-daemon** on all clients.

16.7.3 PPD Files in Various Packages

The YaST printer configuration sets up the queues for CUPS using the PPD files installed in /usr/share/cups/model. To find the suitable PPD files for the printer model, YaST compares the vendor and model determined during hardware detection with the vendors and models in all PPD files. For this purpose, the YaST printer configuration generates a database from the vendor and model information extracted from the PPD files.

The configuration using only PPD files and no other information sources has the advantage that the PPD files in /usr/share/cups/model can be modified freely. For example, if you have PostScript printers the PPD files can be copied directly to /usr/share/cups/model (if they do not already exist in the manufacturer-PPDs or OpenPrintingPPDs-postscript packages) to achieve an optimum configuration for your printers.

Additional PPD files are provided by the following packages:

- gutenprint: the Gutenprint driver and its matching PPDs
- splix: the SpliX driver and its matching PPDs
- OpenPrintingPPDs-ghostscript: PPDs for Ghostscript built-in drivers
- OpenPrintingPPDs-hpijs: PPDs for the HPIJS driver for non-HP printers

16.8 Troubleshooting

The following sections cover some of the most frequently encountered printer hardware and software problems and ways to solve or circumvent these problems. Among the topics covered are GDI printers, PPD files and port configuration. Common network printer problems, defective printouts, and queue handling are also addressed.

16.8.1 Printers without Standard Printer Language Support

These printers do not support any common printer language and can only be addressed with special proprietary control sequences. Therefore they can only work with the operating system versions for which the manufacturer delivers a driver. GDI is a programming interface developed by Microsoft* for graphics devices. Usually the manufacturer delivers drivers only for Windows, and since the Windows driver uses the GDI interface these printers are also called *GDI printers*. The actual problem is not the programming interface, but because these printers can only be addressed with the proprietary printer language of the respective printer model.

Some GDI printers can be switched to operate either in GDI mode or in one of the standard printer languages. See the manual of the printer whether this is possible. Some models require special Windows software to do the switch (note that the Windows printer driver may always switch the printer back into GDI mode when printing from Windows). For other GDI printers there are extension modules for a standard printer language available.

Some manufacturers provide proprietary drivers for their printers. The disadvantage of proprietary printer drivers is that there is no guarantee that these work with the installed print system or that they are suitable for the various hardware platforms. In contrast, printers that support a standard printer language do not depend on a special print system version or a special hardware platform.

Instead of spending time trying to make a proprietary Linux driver work, it may be more cost-effective to purchase a printer which supports a standard printer language (preferably PostScript). This would solve the driver problem once and for all, eliminating the need to install and configure special driver software and obtain driver updates that may be required because of new developments in the print system.

16.8.2 No Suitable PPD File Available for a PostScript Printer

If the manufacturer-PPDs or OpenPrintingPPDs-postscript packages do not contain a suitable PPD file for a PostScript printer, it should be possible to use the PPD file from the driver CD of the printer manufacturer or download a suitable PPD file from the Web page of the printer manufacturer.

If the PPD file is provided as a zip archive (.zip) or a self-extracting zip archive (.exe), unpack it with unzip. First, review the license terms of the PPD file. Then use the cupstestppd utility to check if the PPD file complies with “Adobe PostScript Printer Description File Format Specification, version 4.3.” If the utility returns “FAIL,” the errors in the PPD files are serious and are likely to cause major problems. The problem spots reported by cupstestppd should be eliminated. If necessary, ask the printer manufacturer for a suitable PPD file.

16.8.3 Network Printer Connections

Identifying Network Problems

Connect the printer directly to the computer. For test purposes, configure the printer as a local printer. If this works, the problems are related to the network.

Checking the TCP/IP Network

The TCP/IP network and name resolution must be functional.

Checking a Remote lpd

Use the following command to test if a TCP connection can be established to lpd (port 515) on host:

```
netcat -z host 515 && echo ok || echo failed
```

If the connection to lpd cannot be established, lpd may not be active or there may be basic network problems.

As the user root, use the following command to query a (possibly very long) status report for queue on remote host, provided the respective lpd is active and the host accepts queries:

```
echo -e "\004queue" \  
| netcat -w 2 -p 722 host 515
```

If lpd does not respond, it may not be active or there may be basic network problems. If lpd responds, the response should show why printing is not possible on the queue on host. If you receive a response like that shown in *Example 16.1, "Error Message from lpd"*, the problem is caused by the remote lpd.

EXAMPLE 16.1: ERROR MESSAGE FROM lpd

```
lpd: your host does not have line printer access  
lpd: queue does not exist  
printer: spooling disabled  
printer: printing disabled
```

Checking a Remote cupsd

A CUPS network server can broadcast its queues by default every 30 seconds on UDP port 631. Accordingly, the following command can be used to test whether there is a broadcasting CUPS network server in the network. Make sure to stop your local CUPS daemon before executing the command.

```
netcat -u -l -p 631 & PID=$! ; sleep 40 ; kill $PID
```

If a broadcasting CUPS network server exists, the output appears as shown in *Example 16.2, "Broadcast from the CUPS Network Server"*.

EXAMPLE 16.2: BROADCAST FROM THE CUPS NETWORK SERVER

```
ipp://192.168.2.202:631/printers/queue
```

System z Take into account that IBM z Systems Ethernet devices do not receive broadcasts by default. ◀

The following command can be used to test if a TCP connection can be established to cupsd (port 631) on host:

```
netcat -z host 631 && echo ok || echo failed
```

If the connection to cupsd cannot be established, cupsd may not be active or there may be basic network problems. lpstat -h host -l -t returns a (possibly very long) status report for all queues on host, provided the respective cupsd is active and the host accepts queries.

The next command can be used to test if the queue on host accepts a print job consisting of a single carriage-return character. Nothing should be printed. Possibly, a blank page may be ejected.

```
echo -en "\r" \  
| lp -d queue -h host
```

Troubleshooting a Network Printer or Print Server Box

Spoolers running in a print server box sometimes cause problems when they need to deal with multiple print jobs. Since this is caused by the spooler in the print server box, there no way to resolve this issue. As a work-around, circumvent the spooler in the print server box by addressing the printer connected to the print server box directly with the TCP socket. See [Section 16.4, "Network Printers"](#).

In this way, the print server box is reduced to a converter between the various forms of data transfer (TCP/IP network and local printer connection). To use this method, you need to know the TCP port on the print server box. If the printer is connected to the print server box and turned on, this TCP port can usually be determined with the nmap utility from the nmap package some time after the print server box is powered up. For example, nmap IP-address may deliver the following output for a print server box:

Port	State	Service
23/tcp	open	telnet
80/tcp	open	http
515/tcp	open	printer
631/tcp	open	cups
9100/tcp	open	jetdirect

This output indicates that the printer connected to the print server box can be addressed via TCP socket on port 9100. By default, nmap only checks several commonly known ports listed in /usr/share/nmap/nmap-services. To check all possible ports, use the command nmap -p from_port-to_port IP-address. This may take some time. For further information, refer to the man page of nmap.

Enter a command like

```
echo -en "\rHello\r\f" | netcat -w 1 IP-address port
```

```
cat file | netcat -w 1 IP-address port
```

to send character strings or files directly to the respective port to test if the printer can be addressed on this port.

16.8.4 Defective Printouts without Error Message

For the print system, the print job is completed when the CUPS back-end completes the data transfer to the recipient (printer). If further processing on the recipient fails (for example, if the printer is not able to print the printer-specific data) the print system does not notice this. If the printer cannot print the printer-specific data, select a PPD file that is more suitable for the printer.

16.8.5 Disabled Queues

If the data transfer to the recipient fails entirely after several attempts, the CUPS back-end, such as USB or socket, reports an error to the print system (to cupsd). The back-end determines how many unsuccessful attempts are appropriate until the data transfer is reported as impossible. As further attempts would be in vain, cupsd disables printing for the respective queue. After eliminating the cause of the problem, the system administrator must re-enable printing with the command cupsenable.

16.8.6 CUPS Browsing: Deleting Print Jobs

If a CUPS network server broadcasts its queues to the client hosts via browsing and a suitable local cupsd is active on the client hosts, the client cupsd accepts print jobs from applications and forwards them to the cupsd on the server. When cupsd on the server accepts a print job, it is assigned a new job number. Therefore, the job number on the client host is different from the job number on the server. As a print job is usually forwarded immediately, it cannot be deleted with the job number on the client host. This is because the client cupsd regards the print job as completed when it has been forwarded to the server cupsd.

When it becomes desirable to delete the print job on the server, use a command such as **lpstat -h cups.example.com -o** to determine the job number on the server, provided the server has not already completed the print job (that is, sent it completely to the printer). Using this job number, the print job on the server can be deleted:

```
cancel -h cups.example.com queue-jobnumber
```

16.8.7 Defective Print Jobs and Data Transfer Errors

If you switch the printer off or shut down the computer during the printing process, print jobs remain in the queue. Printing resumes when the computer (or the printer) is switched back on. Defective print jobs must be removed from the queue with **cancel**.

If a print job is defective or an error occurs in the communication between the host and the printer, the printer prints numerous sheets of paper with unintelligible characters, because it cannot process the data correctly. To rectify this situation, follow these steps:

1. To stop printing, remove all paper from ink jet printers or open the paper trays of laser printers. High-quality printers have a button for canceling the current printout.
2. The print job may still be in the queue, because jobs are only removed after they are sent completely to the printer. Use **lpstat -o** or **lpstat -h cups.example.com -o** to check which queue is currently printing. Delete the print job with **cancel queue-jobnumber** or **cancel -h cups.example.com queue-jobnumber**.
3. Some data may still be transferred to the printer even though the print job has been deleted from the queue. Check if a CUPS back-end process is still running for the respective queue and terminate it.
4. Reset the printer completely by switching it off for some time. Then insert the paper and turn on the printer.

16.8.8 Debugging CUPS

Use the following generic procedure to locate problems in CUPS:

1. Set **LogLevel debug** in **/etc/cups/cupsd.conf**.

2. Stop cupsd.
3. Remove /var/log/cups/error_log* to avoid having to search through very large log files.
4. Start cupsd.
5. Repeat the action that led to the problem.
6. Check the messages in /var/log/cups/error_log* to identify the cause of the problem.

16.8.9 For More Information

In-depth information about printing on SUSE Linux is presented in the openSUSE Support Database at <http://en.opensuse.org/Portal:Printing>. Solutions to many specific problems are presented in the SUSE Knowledgebase (<http://www.suse.com/support/>). Locate the relevant articles with a text search for CUPS.

17 The X Window System

The X Window System (X11) is the de facto standard for graphical user interfaces in Unix. X is network-based, enabling applications started on one host to be displayed on another host connected over any kind of network (LAN or Internet). This chapter provides basic information on the X configuration, and background information about the use of fonts in SUSE® Linux Enterprise Server.

Usually, the X Window System needs no configuration. The hardware is dynamically detected during X start-up. The use of `xorg.conf` is therefore deprecated. If you still need to specify custom options to change the way X behaves, you can still do so by modifying configuration files under `/etc/X11/xorg.conf.d/`.



Tip: IBM z Systems: Configuring the Graphical User Interface

IBM z Systems does not have any input or output devices supported by X.Org. Therefore, none of the configuration procedures described in this section apply. More relevant information for IBM z Systems can be found in *Book “Deployment Guide”, Chapter 4 “Installation on IBM z Systems”*.

17.1 Installing and Configuring Fonts

Fonts in Linux can be categorized into two parts:

Outline or Vector Fonts

Contains a mathematical description as drawing instructions about the shape of a glyph. As such, each glyph can be scaled to arbitrary sizes without loss of quality. Before such a font (or glyph) can be used, the mathematical descriptions need to be transformed into a raster (grid). This process is called *font rasterization*. *Font hinting* (embedded inside the font) improves and optimizes the rendering result for a particular size. Rasterization and hinting is done with the FreeType library.

Common formats under Linux are PostScript Type 1 and Type 2, TrueType, and OpenType.


Bitmap or Raster Fonts


Consists of an array of pixels designed for a specific font size. Bitmap fonts are extremely fast and simple to render. However, compared to vector fonts, bitmap fonts cannot be scaled without losing quality. As such, these fonts are usually distributed in different sizes. These days, bitmap fonts are still used in the Linux console and sometimes in terminals. Under Linux, Portable Compiled Format (PCF) or Glyph Bitmap Distribution Format (BDF) are the most common formats.



The appearance of these fonts can be influenced by two main aspects:

- choosing a suitable font family,
- rendering the font with an algorithm that achieves results comfortable for the receiver's eyes.

The last point is only relevant to vector fonts. Although the above two points are highly subjective, some defaults need to be created.

Linux font rendering systems consist of several libraries with different relations. The basic font rendering library is [FreeType](http://www.freetype.org/) (<http://www.freetype.org/>) , which converts font glyphs of supported formats into optimized bitmap glyphs. The rendering process is controlled by an algorithm and its parameters (which may be subject to patent issues).

Every program or library which uses FreeType should consult the [Fontconfig](http://www.fontconfig.org/) (<http://www.fontconfig.org/>)  library. This library gathers font configuration from users and from the system. When a user amends his Fontconfig setting, this change will result in Fontconfig-aware applications.

More sophisticated OpenType shaping needed for scripts such as Arabic, Han or Phags-Pa and other higher level text processing lies on the shoulders of [Harfbuzz](http://www.harfbuzz.org/) (<http://www.harfbuzz.org/>)  or [Pango](http://www.pango.org/) (<http://www.pango.org/>) , to mention some examples.

17.1.1 Showing Installed Fonts

To get an overview about which fonts are installed on your system, ask the commands **rpm** or **fc-list**. Both will give you a good answer, but may return a different list depending on system and user configuration:

rpm

Invoke **rpm** to see which software packages containing fonts are installed on your system:

```
rpm -qa '*fonts*'
```

Every font package should satisfy this expression. However, the command may return some false positives like **fonts-config** (which is neither a font nor does it contain fonts).

fc-list

Invoke **fc-list** to get an overview about what font families can be accessed, whether they are installed on the system or in your home:

```
fc-list ':' family
```



Note: Command **fc-list**

The command **fc-list** is a wrapper to the Fontconfig library. It is possible to query a lot of interesting information from Fontconfig—or, to be more precise, from its cache. See **man 1 fc-list** for more details.

17.1.2 Viewing Fonts

If you want to know what an installed font family looks like, either use the command **ftview** (package **ft2demos**) or visit <http://fontinfo.opensuse.org/>⁷. For example, to display the FreeMono font in 14 point, use **ftview** like this:

```
ftview 14 /usr/share/fonts/truetype/FreeMono.ttf
```

If you need further information, go to <http://fontinfo.opensuse.org/>⁷ to find out which styles (regular, bold, italic, etc.) and languages are supported.

17.1.3 Querying Fonts

To query which font is used when a pattern is given, use the **fc-match** command.

For example, if your pattern contains an already installed font, **fc-match** returns the file name, font family, and the style:

```
tux > fc-match 'Liberation Serif'
LiberationSerif-Regular.ttf: "Liberation Serif" "Regular"
```

If the desired font does not exist on your system, Fontconfig's matching rules take place and try to find the most similar fonts available. This means, your request is substituted:

```
tux > fc-match 'Foo Family'
DejaVuSans.ttf: "DejaVu Sans" "Book"
```

Fontconfig supports *aliases*: a name is substituted with another family name. A typical example are the generic names such as “sans-serif”, “serif”, and “monospace”. These alias names can be substituted by real family names or even a preference list of family names:

```
tux > for font in serif sans mono; do fc-match "$font" ; done
DejaVuSerif.ttf: "DejaVu Serif" "Book"
DejaVuSans.ttf: "DejaVu Sans" "Book"
DejaVuSansMono.ttf: "DejaVu Sans Mono" "Book"
```

The result may vary on your system, depending on which fonts are currently installed.



Note: Similarity Rules according to Fontconfig


Fontconfig *always* returns a real family (if at least one is installed) according to the given request, as similar as possible. “Similarity” depends on Fontconfig's internal metrics and on the user's or administrator's Fontconfig settings.

17.1.4 Installing Fonts

To install a new font there are these major methods:

1. Manually install the font files such as *.ttf or *.otf to a known font directory. If it needs to be system-wide, use the standard directory /usr/share/fonts. For installation in your home directory, use ~/.config/fonts.

If you want to deviate from the standard directories, Fontconfig allows you to choose another one. Let Fontconfig know by using the `<dir>` element, see [Section 17.1.5.2, “Diving into Fontconfig XML”](#) for details.

2. Install fonts using **zypper**. Lots of fonts are already available as a package, be it on your SUSE distribution or in the [M17N:fonts](http://download.opensuse.org/repositories/M17N:/fonts/) (<http://download.opensuse.org/repositories/M17N:/fonts/>)  repository. Add the repository to your list using the following command. For example, to add a repository for SLE 12:

```
sudo zypper ar
    http://download.opensuse.org/repositories/M17N:/fonts/SLE_12/
M17N:fonts.repo
```

To search for your `FONT_FAMILY_NAME` use this command:

```
sudo zypper se 'FONT_FAMILY_NAME*fonts'
```

17.1.5 Configuring the Appearance of Fonts

Depending on the rendering medium, and font size, the result may be unsatisfactory. For example, an average monitor these days has a resolution of 100dpi which makes pixels too big and glyphs look clunky.

There are several algorithms available to deal with low resolutions, such as anti-aliasing (grayscale smoothing), hinting (fitting to the grid), or subpixel rendering (tripling resolution in one direction). These algorithms can also differ from one font format to another.

Important: Patent Issues with Subpixel Rendering

Subpixel rendering is not used in SUSE distributions. Although FreeType2 has support for this algorithm, it is covered by several patents expiring at the end of the year 2019. Therefore, setting subpixel rendering options in Fontconfig has no effect unless the system has a FreeType2 library with subpixel rendering compiled in.

Via Fontconfig, it is possible to select a rendering algorithms for every font individually or for a set of fonts.

17.1.5.1 Configuring Fonts via sysconfig

SUSE Linux Enterprise Server comes with a `sysconfig` layer above `Fontconfig`. This is a good starting point for experimenting with font configuration. To change the default settings, edit the configuration file `/etc/sysconfig/fonts-config`. (or use the YaST `sysconfig` module). After you have edited the file, run `fonts-config`:

```
sudo /usr/sbin/fonts-config
```

Restart the application to make the effect visible. Keep in mind the following issues:

- A few applications do need not to be restarted. For example, Firefox re-reads `Fontconfig` configuration from time to time. Newly created or reloaded tabs get new font configurations later.
- The `fonts-config` script is called automatically after every package installation or removal (if not, it is a bug of the font software package).
- Every `sysconfig` variable can be temporarily overridden by the `fonts-config` command line option. See `fonts-config --help` for details.

There are several `sysconfig` variables which can be altered. See `man 1 fonts-config` or the help page of the YaST `sysconfig` module. The following variables are examples:

Usage of Rendering Algorithms

Consider `FORCE_HINTSTYLE`, `FORCE_AUTOHINT`, `FORCE_BW`, `FORCE_BW_MONOSPACE`, `USE_EMBEDDED_BITMAPS` and `EMBEDDED_BITMAP_LANGAGES`

Preference Lists of Generic Aliases

Use `PREFER_SANS_FAMILIES`, `PREFER_SERIF_FAMILIES`, `PREFER_MONO_FAMILIES` and `SEARCH_METRIC_COMPATIBLE`

The following list provides some configuration examples, sorted from the “most readable” fonts (more contrast) to “most beautiful” (more smoothed).

Bitmap Fonts

Prefer bitmap fonts via the `PREFER_*_FAMILIES` variables. Follow the example in the help section for these variables. Be aware that these fonts are rendered black and white, not smoothed and that bitmap fonts are available in several sizes only. Consider using

```
SEARCH_METRIC_COMPATIBLE="no"
```

to disable metric compatibility-driven family name substitutions.

Scalable Fonts Rendered Black and White

Scalable fonts rendered without antialiasing can result in a similar outcome to bitmap fonts, while maintaining font scalability. Use well hinted fonts like the Liberation families. Unfortunately, there is a lack of well hinted fonts though. Set the following variable to force this method:

```
FORCE_BW="yes"
```

Monospaced Fonts Rendered Black and White

Render monospaced fonts without antialiasing only, otherwise use default settings:

```
FORCE_BW_MONOSPACE="yes"
```

Default Settings

All fonts are rendered with antialiasing. Well hinted fonts will be rendered with the *byte code interpreter* (BCI) and the rest with autohinter (hintstyle=hintslight). Leave all relevant sysconfig variables to the default setting.

CFF Fonts

Use fonts in CFF format. They can be considered also more readable than the default TrueType fonts given the current improvements in FreeType2. Try them out by following the example of PREFER_*_FAMILIES. Possibly make them more dark and bold with:

```
SEARCH_METRIC_COMPATIBLE="no"
```

as they are rendered by hintstyle=hintslight by default. Also consider using:

```
SEARCH_METRIC_COMPATIBLE="no"
```

Autohinter Exclusively

Even for a well hinted font, use FreeType2's autohinter. That can lead to thicker, sometimes fuzzier letter shapes with lower contrast. Set the following variable to activate this:

```
FORCE_AUTOHINTER="yes"
```

Use FORCE_HINTSTYLE to control the level of hinting.

17.1.5.2 Diving into Fontconfig XML

Fontconfig's configuration format is the *eXtensible Markup Language* (XML). These few examples are not a complete reference, but a brief overview. Details and other inspiration can be found in **man 5 fonts-conf** or in `/etc/fonts/conf.d/`.

The central Fontconfig configuration file is `/etc/fonts/fonts.conf`, which—along other work—includes the whole `/etc/fonts/conf.d/` directory. To customize Fontconfig, there are two places where you can insert your changes:

FONTCONFIG CONFIGURATION FILES

1. **System-wide changes.** Edit the file `/etc/fonts/local.conf` (by default, it contains an empty `fontconfig` element).
2. **User-specific changes.** Edit the file `~/.config/fontconfig/fonts.conf`. Place Fontconfig configuration files in the `~/.config/fontconfig/conf.d/` directory.

User-specific changes overwrite any system-wide settings.



Note: Deprecated User Configuration File

The file `~/.fonts.conf` is marked as deprecated and should not be used anymore. Use `~/.config/fontconfig/fonts.conf` instead.

Every configuration file needs to have a `fontconfig` element. As such, the minimal file looks like this:

```
<?xml version="1.0"?>
  <!DOCTYPE fontconfig SYSTEM "fonts.dtd">
  <fontconfig>
    <!-- Insert your changes here -->
  </fontconfig>
```

If the default directories are not enough, insert the `dir` element with the respective directory:

```
<dir>/usr/share/fonts2</dir>
```

Fontconfig searches *recursively* for fonts.

Font-rendering algorithms can be chosen with following Fontconfig snippet (see [Example 17.1, “Specifying Rendering Algorithms”](#)):

EXAMPLE 17.1: SPECIFYING RENDERING ALGORITHMS

```
<match target="font">
  <test name="family">
    <string>FAMILY_NAME</string>
  </test>
  <edit name="antialias" mode="assign">
    <bool>true</bool>
  </edit>
  <edit name="hinting" mode="assign">
    <bool>true</bool>
  </edit>
  <edit name="autohint" mode="assign">
    <bool>false</bool>
  </edit>
  <edit name="hintstyle" mode="assign">
    <const>hintfull</const>
  </edit>
</match>
```

Various properties of fonts can be tested. For example, the `<test>` element can test for the font family (as shown in the example), size interval, spacing, font format, and others. When abandoning `<test>` completely, all `<edit>` elements will be applied to every font (global change).

EXAMPLE 17.2: ALIASES AND FAMILY NAME SUBSTITUTIONS

Rule 1

```
<alias>
  <family>Alegreya SC</family>
  <default>
    <family>serif</family>
  </default>
</alias>
```

Rule 2

```
<alias>
  <family>serif</family>
  <prefer>
```



```
<family>Droid Serif</family>
</prefer>
</alias>
```

Rule 3

```
<alias>
  <family>serif</family>
  <accept>
    <family>STIXGeneral</family>
  </accept>
</alias>
```

The rules from *Example 17.2, “Aliases and Family Name Substitutions”* create a *prioritized family list* (PFL). Depending on the element, different actions are performed:

<default> from *Rule 1*

This rule adds a serif family name *at the end* of the PFL.

<prefer> from *Rule 2*

This rule adds “Droid Serif” *just before* the first occurrence of serif in the PFL, whenever Alegreya SC is present in PFL.

<accept> from *Rule 3*

This rule adds a “STIXGeneral” family name *just after* the first occurrence of the serif family name in the PFL.

Putting this together, when snippets occur in the order *Rule 1 - Rule 2 - Rule 3* and the user requests “Alegreya SC”, then the PFL is created as depicted in *Table 17.1, “Generating PFL from Fontconfig rules”*.

TABLE 17.1: GENERATING PFL FROM FONTCONFIG RULES

Order	Current PFL
Request	<u>Alegreya SC</u>
<i>Rule 1</i>	<u>Alegreya SC</u> , <u>serif</u>
<i>Rule 2</i>	<u>Alegreya SC</u> , <u>Droid Serif</u> , <u>serif</u>
<i>Rule 3</i>	<u>Alegreya SC</u> , <u>Droid Serif</u> , <u>serif</u> , <u>STIXGeneral</u>

In Fontconfig's metrics, the family name has the highest priority over other patterns, like style, size, etc. Fontconfig checks which family is currently installed on the system. If “Alegreya SC” is installed, then Fontconfig returns it. If not, it asks for “Droid Serif”, etc.

Be careful. When the order of Fontconfig snippets is changed, Fontconfig can return different results, as depicted in *Table 17.2, “Results from Generating PFL from Fontconfig Rules with Changed Order”*.

TABLE 17.2: RESULTS FROM GENERATING PFL FROM FONTCONFIG RULES WITH CHANGED ORDER

Order	Current PFL	Note
Request	<u>Alegreya SC</u>	Same request performed.
<i>Rule 2</i>	<u>Alegreya SC</u>	<u>serif</u> not in FPL, nothing is substituted
<i>Rule 3</i>	<u>Alegreya SC</u>	<u>serif</u> not in FPL, nothing is substituted
<i>Rule 1</i>	<u>Alegreya SC</u> , <u>serif</u>	<u>Alegreya SC</u> present in FPL, substitution is performed



Note: Implication.

Think of the <default> alias as a classification or inclusion of this group (if not installed). As the example shows, <default> should always precede the <prefer> and <accept> aliases of that group.

<default> classification is not limited to the generic aliases serif, sans-serif and monospace. See /usr/share/fontconfig/conf.avail/30-metric-aliases.conf for a complex example.

The following Fontconfig snippet in *Example 17.3, “Aliases and Family Name Substitutions”* creates a serif group. Every family in this group could substitute others when a former font is not installed.

EXAMPLE 17.3: ALIASES AND FAMILY NAME SUBSTITUTIONS

```
<alias>
  <family>Alegreya SC</family>
```

```

<default>
  <family>serif</family>
</default>
</alias>
<alias>
  <family>Droid Serif</family>
  <default>
    <family>serif</family>
  </default>
</alias>
<alias>
  <family>STIXGeneral</family>
  <default>
    <family>serif</family>
  </default>
</alias>
<alias>
  <family>serif</family>
  <accept>
    <family>Droid Serif</family>
    <family>STIXGeneral</family>
    <family>Alegreya SC</family>
  </accept>
</alias>

```

Priority is given by the order in the `<accept>` alias. Similarly, stronger `<prefer>` aliases can be used.

Example 17.2, “Aliases and Family Name Substitutions” is expanded by Example 17.4, “Aliases and Family Names Substitutions”.

EXAMPLE 17.4: ALIASES AND FAMILY NAMES SUBSTITUTIONS

Rule 4

```

<alias>
  <family>serif</family>
  <accept>
    <family>Liberation Serif</family>
  </accept>
</alias>

```

Rule 5

```

<alias>

```

```

<family>serif</family>
<prefer>
  <family>DejaVu Serif</family>
</prefer>
</alias>

```

The expanded configuration from *Example 17.4, “Aliases and Family Names Substitutions”* would lead to the following PFL evolution:

TABLE 17.3: RESULTS FROM GENERATING PFL FROM FONTCONFIG RULES

Order	Current PFL
Request	<u>Alegreya SC</u>
<i>Rule 1</i>	<u>Alegreya SC</u> , <u>serif</u>
<i>Rule 2</i>	<u>Alegreya SC</u> , <u>Droid Serif</u> , <u>serif</u>
<i>Rule 3</i>	<u>Alegreya SC</u> , <u>Droid Serif</u> , <u>serif</u> , <u>STIXGeneral</u>
<i>Rule 4</i>	<u>Alegreya SC</u> , <u>Droid Serif</u> , <u>serif</u> , <u>Liberation Serif</u> , <u>STIX-General</u>
<i>Rule 5</i>	<u>Alegreya SC</u> , <u>Droid Serif</u> , <u>DejaVu Serif</u> , <u>serif</u> , <u>Liberation Serif</u> , <u>STIXGeneral</u>



Note: Implications.

- In case multiple <accept> declarations for the same generic name exist, the declaration that is parsed last “wins”. If possible, do not use <accept> **after** user (/ etc/fonts/conf.d/*-user.conf) when creating a system-wide configuration.
- In case multiple <prefer> declarations for the same generic name exist, the declaration that is parsed last “wins”. If possible, do not use <prefer> **before** user in the system-wide configuration.
- Every <prefer> declaration overwrites <accept> declarations for the same generic name. If the administrator wants to give the user free rein to use even <accept> and not only <prefer>, the administrator should not use <prefer> in the

system-wide configuration. On the other hand, users mostly use `<prefer>`, so that should not be detrimental and we see the use of `<prefer>` also in system wide configurations.

17.2 For More Information

Install the packages `xorg-docs` to get more in-depth information about X11. `man 5 xorg.conf` tells you more about the format of the manual configuration (if needed). More information on the X11 development can be found on the project's home page at <http://www.x.org>.

Drivers are found in `xf86-video-*` packages, for example `xf86-video-nv`. Many of the drivers delivered with these packages are described in detail in the related manual page. For example, if you use the `nv` driver, find more information about this driver in `man 4 nv`.

Information about third-party drivers should be available in `/usr/share/doc/packages/<package_name>`. For example, the documentation of `x11-video-nvidiaG03` is available in `/usr/share/doc/packages/x11-video-nvidiaG03` after the package was installed.

18 Accessing File Systems with FUSE

FUSE is the acronym for *file system in user space*. This means you can configure and mount a file system as an unprivileged user. Normally, you need to be root for this task. FUSE alone is a kernel module. Combined with plug-ins, it allows you to extend FUSE to access almost all file systems like remote SSH connections, ISO images, and more.

18.1 Configuring FUSE

Before you can use FUSE, you need to install the package fuse. Depending which file system you want to use, you need additional plug-ins available as separate packages.

Generally you do not need to configure FUSE. However, it is a good idea to create a directory where all your mount points are combined. For example, you can create a directory ~/mounts and insert your subdirectories for your different file systems there.

18.2 Mounting an NTFS Partition

NTFS, the *New Technology File System*, is the default file system of Windows. To mount a Windows partition as a normal user, proceed as follows:

1. Become root and install the package ntfs-3g.
2. Create a directory that is to be used as a mount point, for example ~/mounts/windows.
3. Find out which Windows partition you need. Use YaST and start the partitioner module to see which partition belongs to Windows, but do not modify anything. Alternatively, become root and execute /sbin/fdisk -l. Look for partitions with a partition type of HPFS/NTFS.
4. Mount the partition in read-write mode. Replace the placeholder DEVICE with your respective Windows partition:

```
ntfs-3g /dev/DEVICE MOUNT POINT
```

To use your Windows partition in read-only mode, append `-o ro`:

```
ntfs-3g /dev/DEVICE MOUNT POINT -o ro
```

The command `ntfs-3g` uses the current user (UID) and group (GID) to mount the given device. If you want to set the write permissions to a different user, use the command `id USER` to get the output of the UID and GID values. Set it with:

```
id tux
uid=1000(tux) gid=100(users) groups=100(users),16(dialout),33(video)
ntfs-3g /dev/DEVICE MOUNT POINT -o uid=1000,gid=100
```

Find additional options in the man page.

To unmount the resource, run `fusermount -u MOUNT POINT`.

18.3 For More Information

See the home page <http://fuse.sourceforge.net> of FUSE for more information.

19 Dynamic Kernel Device Management with udev

The kernel can add or remove almost any device in a running system. Changes in the device state (whether a device is plugged in or removed) need to be propagated to user space. Devices need to be configured as soon as they are plugged in and recognized. Users of a certain device need to be informed about any changes in this device's recognized state. udev provides the needed infrastructure to dynamically maintain the device node files and symbolic links in the /dev directory. udev rules provide a way to plug external tools into the kernel device event processing. This enables you to customize udev device handling by, for example, adding certain scripts to execute as part of kernel device handling, or request and import additional data to evaluate during device handling.

19.1 The /dev Directory

The device nodes in the /dev directory provide access to the corresponding kernel devices. With udev, the /dev directory reflects the current state of the kernel. Every kernel device has one corresponding device file. If a device is disconnected from the system, the device node is removed.

The content of the /dev directory is kept on a temporary file system and all files are rendered at every system start-up. Manually created or modified files do not, by design, survive a reboot. Static files and directories that should always be in the /dev directory regardless of the state of the corresponding kernel device can be created with systemd-tmpfiles. The configuration files are found in /usr/lib/tmpfiles.d/ and /etc/tmpfiles.d/; for more information, see the systemd-tmpfiles(8) man page.

19.2 Kernel uevents and udev

The required device information is exported by the sysfs file system. For every device the kernel has detected and initialized, a directory with the device name is created. It contains attribute files with device-specific properties.

Every time a device is added or removed, the kernel sends a uevent to notify udev of the change. The udev daemon reads and parses all provided rules from the /etc/udev/rules.d/*.rules files once at start-up and keeps them in memory. If rules files are changed, added or removed, the daemon can reload the in-memory representation of all rules with the command **udevadm control reload_rules**. For more details on udev rules and their syntax, refer to [Section 19.6, “Influencing Kernel Device Event Handling with udev Rules”](#).

Every received event is matched against the set of provided rules. The rules can add or change event environment keys, request a specific name for the device node to create, add symbolic links pointing to the node or add programs to run after the device node is created. The driver core uevents are received from a kernel netlink socket.

19.3 Drivers, Kernel Modules and Devices

The kernel bus drivers probe for devices. For every detected device, the kernel creates an internal device structure while the driver core sends a uevent to the udev daemon. Bus devices identify themselves by a specially-formatted ID, which tells what kind of device it is. Usually these IDs consist of vendor and product ID and other subsystem-specific values. Every bus has its own scheme for these IDs, called MODALIAS. The kernel takes the device information, composes a MODALIAS ID string from it and sends that string along with the event. For a USB mouse, it looks like this:

```
MODALIAS=usb:v046DpC03Ed2000dc00dsc00dp00ic03isc01ip02
```

Every device driver carries a list of known aliases for devices it can handle. The list is contained in the kernel module file itself. The program `depmod` reads the ID lists and creates the file modules.alias in the kernel's /lib/modules directory for all currently available modules. With this infrastructure, module loading is as easy as calling **modprobe** for every event that carries a MODALIAS key. If **modprobe \$MODALIAS** is called, it matches the device alias composed for the device with the aliases provided by the modules. If a matching entry is found, that module is loaded. All this is automatically triggered by udev.

19.4 Booting and Initial Device Setup

All device events happening during the boot process before the `udev` daemon is running are lost, because the infrastructure to handle these events resides on the root file system and is not available at that time. To cover that loss, the kernel provides a `uevent` file located in the device directory of every device in the `sysfs` file system. By writing `add` to that file, the kernel resends the same event as the one lost during boot. A simple loop over all `uevent` files in `/sys` triggers all events again to create the device nodes and perform device setup.

As an example, a USB mouse present during boot may not be initialized by the early boot logic, because the driver is not available at that time. The event for the device discovery was lost and failed to find a kernel module for the device. Instead of manually searching for possibly connected devices, `udev` requests all device events from the kernel after the root file system is available, so the event for the USB mouse device runs again. Now it finds the kernel module on the mounted root file system and the USB mouse can be initialized.

From user space, there is no visible difference between a device coldplug sequence and a device discovery during runtime. In both cases, the same rules are used to match and the same configured programs are run.

19.5 Monitoring the Running udev Daemon

The program `udevadm monitor` can be used to visualize the driver core events and the timing of the `udev` event processes.

```
UEVENT[1185238505.276660] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1 (usb)
UDEV [1185238505.279198] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1 (usb)
UEVENT[1185238505.279527] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0
(usb)
UDEV [1185238505.285573] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0
(usb)
UEVENT[1185238505.298878] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/
input/input10 (input)
UDEV [1185238505.305026] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/
input/input10 (input)
UEVENT[1185238505.305442] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/
input/input10/mouse2 (input)
UEVENT[1185238505.306440] add /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/
input/input10/event4 (input)
```

```
UDEV [1185238505.325384] add    /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/
input/input10/event4 (input)
UDEV [1185238505.342257] add    /devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/
input/input10/mouse2 (input)
```

The UEVENT lines show the events the kernel has sent over netlink. The UDEV lines show the finished udev event handlers. The timing is printed in microseconds. The time between UEVENT and UDEV is the time udev took to process this event or the udev daemon has delayed its execution to synchronize this event with related and already running events. For example, events for hard disk partitions always wait for the main disk device event to finish, because the partition events may rely on the data that the main disk event has queried from the hardware.

udevadm monitor --env shows the complete event environment:

```
ACTION=add
DEVPATH=/devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/input/input10
SUBSYSTEM=input
SEQNUM=1181
NAME="Logitech USB-PS/2 Optical Mouse"
PHYS="usb-0000:00:1d.2-1/input0"
UNIQ=""
EV=7
KEY=70000 0 0 0 0
REL=103
MODALIAS=input:b0003v046DpC03Ee0110-e0,1,2,k110,111,112,r0,1,8,amlsfw
```

udev also sends messages to syslog. The default syslog priority that controls which messages are sent to syslog is specified in the udev configuration file /etc/udev/udev.conf. The log priority of the running daemon can be changed with **udevadm control log_priority= level/number**.

19.6 Influencing Kernel Device Event Handling with udev Rules

A udev rule can match any property the kernel adds to the event itself or any information that the kernel exports to sysfs. The rule can also request additional information from external programs. Every event is matched against all provided rules. All rules are located in the /etc/udev/rules.d directory.

Every line in the rules file contains at least one key value pair. There are two kinds of keys, match and assignment keys. If all match keys match their values, the rule is applied and the assignment keys are assigned the specified value. A matching rule may specify the name of the device node, add symbolic links pointing to the node or run a specified program as part of the event handling. If no matching rule is found, the default device node name is used to create the device node. Detailed information about the rule syntax and the provided keys to match or import data are described in the `udev` man page. The following example rules provide a basic introduction to `udev` rule syntax. The example rules are all taken from the `udev` default rule set that is located under `/etc/udev/rules.d/50-udev-default.rules`.

EXAMPLE 19.1: EXAMPLE `udev` RULES

```
# console
KERNEL=="console", MODE="0600", OPTIONS="last_rule"

# serial devices
KERNEL=="ttyUSB*", ATTRS{product}=="[Pp]alm*Handheld*", SYMLINK+="pilot"

# printer
SUBSYSTEM=="usb", KERNEL=="lp*", NAME="usb/%k", SYMLINK+="usb%k", GROUP="lp"

# kernel firmware loader
SUBSYSTEM=="firmware", ACTION=="add", RUN+="firmware.sh"
```

The `console` rule consists of three keys: one match key (`KERNEL`) and two assign keys (`MODE`, `OPTIONS`). The `KERNEL` match rule searches the device list for any items of the type `console`. Only exact matches are valid and trigger this rule to be executed. The `MODE` key assigns special permissions to the device node, in this case, read and write permissions to the owner of this device only. The `OPTIONS` key makes this rule the last rule to be applied to any device of this type. Any later rule matching this particular device type does not have any effect.

The `serial devices` rule is not available in `50-udev-default.rules` anymore, but it is still worth considering. It consists of two match keys (`KERNEL` and `ATTRS`) and one assign key (`SYMLINK`). The `KERNEL` key searches for all devices of the `ttyUSB` type. Using the `*` wild card, this key matches several of these devices. The second match key, `ATTRS`, checks whether the `product` attribute file in `sysfs` for any `ttyUSB` device contains a certain string. The assign key (`SYMLINK`) triggers the addition of a symbolic link to this device under `/dev/pilot`. The operator used in this key (`+=`) tells `udev` to additionally perform this action, even if previous or later rules add other symbolic links. As this rule contains two match keys, it is only applied if both conditions are met.

The `printer` rule deals with USB printers and contains two match keys which must both apply to get the entire rule applied (`SUBSYSTEM` and `KERNEL`). Three assign keys deal with the naming for this device type (`NAME`), the creation of symbolic device links (`SYMLINK`) and the group membership for this device type (`GROUP`). Using the `*` wild card in the `KERNEL` key makes it match several `lp` printer devices. Substitutions are used in both, the `NAME` and the `SYMLINK` keys to extend these strings by the internal device name. For example, the symbolic link to the first `lp` USB printer would read `/dev/usb/lp0`.

The `kernel firmware loader` rule makes `udev` load additional firmware by an external helper script during runtime. The `SUBSYSTEM` match key searches for the `firmware` subsystem. The `ACTION` key checks whether any device belonging to the `firmware` subsystem has been added. The `RUN+=` key triggers the execution of the `firmware.sh` script to locate the firmware that is to be loaded.

Some general characteristics are common to all rules:

- Each rule consists of one or more key value pairs separated by a comma.
- A key's operation is determined by the operator. `udev` rules support several different operators.
- Each given value must be enclosed by quotation marks.
- Each line of the rules file represents one rule. If a rule is longer than one line, use `\` to join the different lines as you would do in shell syntax.
- `udev` rules support a shell-style pattern that matches the `*`, `?`, and `[]` patterns.
- `udev` rules support substitutions.

19.6.1 Using Operators in udev Rules

Creating keys you can choose from several operators, depending on the type of key you want to create. Match keys will normally be used to find a value that either matches or explicitly mismatches the search value. Match keys contain either of the following operators:

`==`

Compare for equality. If the key contains a search pattern, all results matching this pattern are valid.

!=

Compare for non-equality. If the key contains a search pattern, all results matching this pattern are valid.

Any of the following operators can be used with assign keys:

=

Assign a value to a key. If the key previously consisted of a list of values, the key resets and only the single value is assigned.

+=

Add a value to a key that contains a list of entries.

:=

Assign a final value. Disallow any later change by later rules.

19.6.2 Using Substitutions in udev Rules

udev rules support the use of placeholders and substitutions. Use them in a similar fashion as you would do in any other scripts. The following substitutions can be used with udev rules:

%r, \$root

The device directory, /dev by default.

%p, \$devpath

The value of DEVPATH.

%k, \$kernel

The value of KERNEL or the internal device name.

%n, \$number

The device number.

%N, \$tempnode

The temporary name of the device file.

%M, \$major

The major number of the device.

%m, \$minor

The minor number of the device.

%s{attribute}, \$attr{attribute}

The value of a sysfs attribute (specified by attribute).

%E{variable}, \$attr{variable}

The value of an environment variable (specified by variable).

%c, \$result

The output of PROGRAM.

%%

The % character.

\$\$

The \$ character.

19.6.3 Using udev Match Keys

Match keys describe conditions that must be met before a udev rule can be applied. The following match keys are available:

ACTION

The name of the event action, for example, add or remove when adding or removing a device.

DEVPATH

The device path of the event device, for example, DEVPATH=/bus/pci/drivers/ipw3945 to search for all events related to the ipw3945 driver.

KERNEL

The internal (kernel) name of the event device.

SUBSYSTEM

The subsystem of the event device, for example, SUBSYSTEM=usb for all events related to USB devices.

ATTR{filename}

sysfs attributes of the event device. To match a string contained in the vendor attribute file name, you could use ATTR{vendor}=="On[sS]tream", for example.

KERNELS

Let udev search the device path upwards for a matching device name.

SUBSYSTEMS

Let udev search the device path upwards for a matching device subsystem name.

DRIVERS

Let udev search the device path upwards for a matching device driver name.

ATTRS{filename}

Let udev search the device path upwards for a device with matching sysfs attribute values.

ENV{key}

The value of an environment variable, for example, ENV{ID_BUS}="ieee1394 to search for all events related to the FireWire bus ID.

PROGRAM

Let udev execute an external program. To be successful, the program must return with exit code zero. The program's output, printed to STDOUT, is available to the RESULT key.

RESULT

Match the output string of the last PROGRAM call. Either include this key in the same rule as the PROGRAM key or in a later one.

19.6.4 Using udev Assign Keys

In contrast to the match keys described above, assign keys do not describe conditions that must be met. They assign values, names and actions to the device nodes maintained by udev.

NAME

The name of the device node to be created. After a rule has set a node name, all other rules with a NAME key for this node are ignored.

SYMLINK

The name of a symbolic link related to the node to be created. Multiple matching rules can add symbolic links to be created with the device node. You can also specify multiple symbolic links for one node in one rule using the space character to separate the symbolic link names.

OWNER, GROUP, MODE

The permissions for the new device node. Values specified here overwrite anything that has been compiled in.

ATTR{key}

Specify a value to be written to a sysfs attribute of the event device. If the == operator is used, this key is also used to match against the value of a sysfs attribute.

ENV{key}

Tell udev to export a variable to the environment. If the == operator is used, this key is also used to match against an environment variable.

RUN

Tell udev to add a program to the list of programs to be executed for this device. Keep in mind to restrict this to very short tasks to avoid blocking further events for this device.

LABEL

Add a label where a GOTO can jump to.

GOTO

Tell udev to skip a number of rules and continue with the one that carries the label referenced by the GOTO key.

IMPORT{type}

Load variables into the event environment such as the output of an external program. udev imports variables of several types. If no type is specified, udev tries to determine the type itself based on the executable bit of the file permissions.

- program tells udev to execute an external program and import its output.
- file tells udev to import a text file.
- parent tells udev to import the stored keys from the parent device.

WAIT_FOR_SYSFS

Tells udev to wait for the specified sysfs file to be created for a certain device. For example, WAIT_FOR_SYSFS="ioerr_cnt" informs udev to wait until the ioerr_cnt file has been created.

OPTIONS

The OPTION key may have several values:

- last_rule tells udev to ignore all later rules.
- ignore_device tells udev to ignore this event completely.

- `ignore_remove` tells `udev` to ignore all later remove events for the device.
- `all_partitions` tells `udev` to create device nodes for all available partitions on a block device.

19.7 Persistent Device Naming

The dynamic device directory and the `udev` rules infrastructure make it possible to provide stable names for all disk devices—regardless of their order of recognition or the connection used for the device. Every appropriate block device the kernel creates is examined by tools with special knowledge about certain buses, drive types or file systems. Along with the dynamic kernel-provided device node name, `udev` maintains classes of persistent symbolic links pointing to the device:

```
/dev/disk
|-- by-id
|  |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B -> ../../sda
|  |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B-part1 -> ../../sda1
|  |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B-part6 -> ../../sda6
|  |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B-part7 -> ../../sda7
|  |-- usb-Generic_STORAGE_DEVICE_02773 -> ../../sdd
|  `-- usb-Generic_STORAGE_DEVICE_02773-part1 -> ../../sdd1
|-- by-label
|  |-- Photos -> ../../sdd1
|  |-- SUSE10 -> ../../sda7
|  `-- devel -> ../../sda6
|-- by-path
|  |-- pci-0000:00:1f.2-scsi-0:0:0:0 -> ../../sda
|  |-- pci-0000:00:1f.2-scsi-0:0:0:0-part1 -> ../../sda1
|  |-- pci-0000:00:1f.2-scsi-0:0:0:0-part6 -> ../../sda6
|  |-- pci-0000:00:1f.2-scsi-0:0:0:0-part7 -> ../../sda7
|  |-- pci-0000:00:1f.2-scsi-1:0:0:0 -> ../../sr0
|  |-- usb-02773:0:0:2 -> ../../sdd
|  |-- usb-02773:0:0:2-part1 -> ../../sdd1
`-- by-uuid
   |-- 159a47a4-e6e6-40be-a757-a629991479ae -> ../../sda7
   |-- 3e999973-00c9-4917-9442-b7633bd95b9e -> ../../sda6
   `-- 4210-8F8C -> ../../sdd1
```

19.8 Files used by udev

/sys/*

Virtual file system provided by the Linux kernel, exporting all currently known devices. This information is used by udev to create device nodes in /dev

/dev/*

Dynamically created device nodes and static content created with systemd-tmpfiles; for more information, see the systemd-tmpfiles(8) man page.

The following files and directories contain the crucial elements of the udev infrastructure:

/etc/udev/udev.conf

Main udev configuration file.

/etc/udev/rules.d/*

udev event matching rules.

/usr/lib/tmpfiles.d/ and /etc/tmpfiles.d/

Responsible for static /dev content.

/usr/lib/udev/*

Helper programs called from udev rules.

19.9 For More Information

For more information about the udev infrastructure, refer to the following man pages:

udev

General information about udev, keys, rules and other important configuration issues.

udevadm

udevadm can be used to control the runtime behavior of udev, request kernel events, manage the event queue and provide simple debugging mechanisms.

udev

Information about the udev event managing daemon.

20 Live Patching the Linux Kernel Using kGraft

This document describes the basic principles of the kGraft live patching technology and provides usage guidelines for the SLE Live Patching service.

kGraft is a live patching technology for runtime patching of the Linux kernel, without stopping the kernel. This maximizes system uptime, and thus system availability, which is important for mission-critical systems. By allowing dynamic patching of the kernel, the technology also encourages users to install critical security updates without deferring them to a scheduled downtime.

A kGraft patch is a kernel module, intended for replacing whole functions in the kernel. kGraft primarily offers in-kernel infrastructure for integration of the patched code with base kernel code at runtime.

SLE Live Patching is a service provided on top of regular SUSE Linux Enterprise Server maintenance. kGraft patches distributed through SLE Live Patching supplement regular SLES maintenance updates. Common update stack and procedures can be used for SLE Live Patching deployment.

20.1 Advantages of kGraft

Live kernel patching using kGraft is especially useful for quick response in emergencies (when serious vulnerabilities are known and should be fixed when possible or there are serious system stability issues with a known fix). It is not used for scheduled updates where time is not critical. Typical use cases for kGraft include systems like memory databases with huge amounts of RAM, where boot-up times of 15 minutes or more are no exception, large simulations that need weeks or months without a restart, or infrastructure building blocks providing continuous service to a many consumers.

The main advantage of kGraft is that it never requires stopping the kernel, not even for a short time period.

A kGraft patch is a `.ko` kernel module in a KMP RPM package. It is inserted into the kernel using the `insmod` command when the RPM package is installed or updated. kGraft replaces whole functions in the kernel, even if they are being executed. An updated kGraft module can replace an existing patch if necessary.

kGraft is also lean—it contains only a small amount of code, because it leverages other standard Linux technologies.

20.2 Low-level Function of kGraft

kGraft uses the ftrace infrastructure to perform patching. The following describes the implementation on the AMD64/Intel 64 architecture.

To patch a kernel function, kGraft needs some space at the start of the function to insert a jump to a new function. This space is allocated during kernel compilation by GCC with function profiling turned on. In particular, a 5-byte call instruction is injected to the start of kernel functions. When such instrumented kernel is booting, profiling calls are replaced by 5-byte NOP (no operation) instructions.

After patching starts, the first byte is replaced by the INT3 (breakpoint) instruction. This ensures atomicity of the 5-byte instruction replacement. The other four bytes are replaced by the address to the new function. Finally, the first byte is replaced by the JMP (long jump) opcode.

Inter-processor non-maskable interrupts (IPI NMI) are used throughout the process to flush speculative decoding queues of other CPUs in the system. This allows switching to the new function without ever stopping the kernel, not even for a very short moment. The interruptions by IPI NMIs can be measured in microseconds and are not considered service interruptions as they happen while the kernel is running in any case.

Callers are never patched. Instead, the callee's NOPs are replaced by a JMP to the new function. JMP instructions remain forever. This takes care of function pointers, including in structures, and does not require saving any old data for the possibility of un-patching.

However, these steps alone would not be good enough: since the functions would be replaced non-atomically, a new fixed function in one part of the kernel could still be calling an old function elsewhere or vice versa. If the semantics of the function interfaces changed in the patch, chaos would ensue.

Thus, until all functions are replaced, kGraft uses an approach based on trampolines and similar to RCU (read-copy-update), to ensure a consistent view of the world to each user space thread, kernel thread and kernel interrupt. A per-thread flag is set on each kernel entry and exit. This way, an old function would always call another old function and a new function always a new one. Once all processes have the "new universe" flag set, patching is complete, trampolines can be removed and the code can operate at full speed without performance impact other than an extra-long jump for each patched function.

20.3 Installing kGraft Patches

This section describes the activation of the SUSE Linux Enterprise Live Patching extension and the installation of kGraft patches.

20.3.1 Activation of SLE Live Patching

To activate SLE Live Patching on your system, follow these steps:

1. If your SLES system is not yet registered, register it. Registration can be done during the system installation or later using the YaST *Product Registration* module (**yast2 registration**). After registration, click *Yes* to see the list of available online updates.
If your SLES system is already registered, but SLE Live Patching is not yet activated, open the YaST *Product Registration* module (**yast2 registration**) and click *Select Extensions*.
2. Select *SUSE Linux Enterprise Live Patching 12* in the list of available extensions and click *Next*.
3. Confirm the license terms and click *Next*.
4. Enter the SLE Live Patching registration code and click *Next*.
5. Check the *Installation Summary* and selected *Patterns*. The pattern Live Patching should be selected for installation.
6. Click *Accept* to complete the installation. This will install the base kGraft components on your system together with the initial live patch.

20.3.2 Updating System

1. SLE Live Patching updates are distributed in a form that allows using standard SLE update stack for patch application. The initial live patch can be updated using **zypper patch**, YaST Online Update or equivalent method.
2. The kernel is patched automatically during the package installation. However, invocations of the old kernel functions are not completely eliminated until all sleeping processes wake up and get out of the way. This can take a considerable amount of time. Despite this,

sleeping processes that use the old kernel functions are not considered a security issue. Nevertheless, in the current version of kGraft, it is not possible to apply another kGraft patch until all processes cross the kernel-user space boundary to use patched functions from the previous patch.

To see the global status of patching, check the flag in `/sys/kernel/kgraft/in_progress`. The value `1` signifies the existence of sleeping processes that still need to be woken (the patching is still in progress). The value `0` signifies that all processes are using solely the patched functions and patching has finished already. Alternatively, use the `kgr status` command to obtain the same information.

The flag can be checked on a per-process basis too. Check the number in `/proc/process_number/kg_r_in_progress` for each process individually. Again, the value `1` signifies sleeping process that still needs to be woken. Alternatively, use the `kgr blocking` command to output the list of sleeping processes.

20.4 Removing a kGraft Patch

To remove a kGraft patch, use the following procedure:

1. First remove the patch itself using Zypper:

```
zypper rm kg_rft-patch-3_12_32-25-default
```

2. Then reboot the machine.

20.5 Stuck Kernel Execution Threads

Kernel threads need to be prepared to handle kGraft. Third-party software may not be completely ready for kGraft adoption and its kernel modules may spawn kernel execution threads. These threads will block the patching process indefinitely. As an emergency measure kGraft offers the possibility to force finishing of the patching process without waiting for all execution threads to cross the safety checkpoint. This can be achieved by writing `0` into `/sys/kernel/kg_rft/in_progress`. Consult SUSE Support before performing this procedure.

20.6 The **kgr** Tool

Several kGraft management tasks can be simplified with the **kgr** tool. The available commands are:

kgr status

Displays the overall status of kGraft patching (ready or in_progress).

kgr patches

Displays the list of loaded kGraft patches.

kgr blocking

Lists processes that are preventing kGraft patching from finishing. By default only the PIDs are listed. Specifying -v prints command lines if available. Another -v displays also stack traces.


For detailed information, see man kgr.

20.7 Scope of kGraft Technology

kGraft is based on replacing functions. Data structure alteration can be accomplished only indirectly with kGraft. As a result, changes to kernel data structure require special care and, if the change is too large, rebooting might be required. kGraft also might not be able to handle situations where one compiler is used to compile the old kernel and another compiler is used for compiling the patch.

Because of the way kGraft works, support for third-party modules that are spawning kernel threads is limited.

20.8 Scope of SLE Live Patching

Fixes for SUSE CVSS (Common Vulnerability Scoring System) level 6+ vulnerabilities and bug fixes related to system stability or data corruption will be shipped in the scope of SLE Live Patching. It might not be possible to produce a live patch for all kinds of fixes fulfilling the above criteria. SUSE reserves the right to skip fixes where production of a kernel live patch is unviable because of technical reasons. For more information on CVSS, which is the base for the SUSE CVSS rating, see <http://nvd.nist.gov/cvss.cfm/> .

20.9 Interaction with the Support Processes

While resolving a technical difficulty with SUSE Support, you may receive a so-called Program Temporary Fix (PTF). PTFs may be issued for various packages including those forming the base of SLE Live Patching.

kGraft PTFs complying with the conditions described in the previous section can be installed as usual and SUSE will ensure that the system in question does not need to be rebooted and that future live updates are applied cleanly.

PTFs issued for the base kernel disrupt the live patching process. First, installing the PTF kernel means a reboot as the kernel cannot be replaced as a whole at runtime. Second, another reboot is needed to replace the PTF with any regular maintenance updates for which the live patches are issued.

PTFs for other packages in SLE Live Patching can be treated like regular PTFs with the usual guarantees.

21 Special System Features

This chapter starts with information about various software packages, the virtual consoles and the keyboard layout. We talk about software components like `bash`, `cron` and `logrotate`, because they were changed or enhanced during the last release cycles. Even if they are small or considered of minor importance, users should change their default behavior, because these components are often closely coupled with the system. The chapter concludes with a section about language and country-specific settings (I18N and L10N).

21.1 Information about Special Software Packages

The programs `bash`, `cron`, `logrotate`, `locate`, `ulimit` and `free` are very important for system administrators and many users. Man pages and info pages are two useful sources of information about commands, but both are not always available. GNU Emacs is a popular and very configurable text editor.

21.1.1 The `bash` Package and `/etc/profile`

Bash is the default system shell. When used as a login shell, it reads several initialization files. Bash processes them in the order they appear in this list:

1. `/etc/profile`
2. `~/.profile`
3. `/etc/bash.bashrc`
4. `~/.bashrc`

Make custom settings in `~/.profile` or `~/.bashrc`. To ensure the correct processing of these files, it is necessary to copy the basic settings from `/etc/skel/.profile` or `/etc/skel/.bashrc` into the home directory of the user. It is recommended to copy the settings from `/etc/skel` after an update. Execute the following shell commands to prevent the loss of personal adjustments:

```
mv ~/.bashrc ~/.bashrc.old
```

```
cp /etc/skel/.bashrc ~/.bashrc
mv ~/.profile ~/.profile.old
cp /etc/skel/.profile ~/.profile
```

Then copy personal adjustments back from the *.old files.

21.1.2 The cron Package

If you want to run commands regularly and automatically in the background at predefined times, cron is the tool to use. cron is driven by specially formatted time tables. Some come with the system and users can write their own tables if needed.

The cron tables are located in /var/spool/cron/tabs. /etc/crontab serves as a systemwide cron table. Enter the user name to run the command directly after the time table and before the command. In *Example 21.1, "Entry in /etc/crontab"*, root is entered. Package-specific tables, located in /etc/cron.d, have the same format. See the cron man page (man cron).

EXAMPLE 21.1: ENTRY IN /ETC/CRONTAB

```
1-59/5 * * * * root test -x /usr/sbin/atrun && /usr/sbin/atrun
```

You cannot edit /etc/crontab by calling the command crontab -e. This file must be loaded directly into an editor, then modified and saved.

A number of packages install shell scripts to the directories /etc/cron.hourly, /etc/cron.daily, /etc/cron.weekly and /etc/cron.monthly, whose execution is controlled by /usr/lib/cron/run-crons. /usr/lib/cron/run-crons is run every 15 minutes from the main table (/etc/crontab). This guarantees that processes that may have been neglected can be run at the proper time.

To run the hourly, daily or other periodic maintenance scripts at custom times, remove the time stamp files regularly using /etc/crontab entries (see *Example 21.2, "/etc/crontab: Remove Time Stamp Files"*, which removes the hourly one before every full hour, the daily one once a day at 2:14 a.m., etc.).

EXAMPLE 21.2: /ETC/CRONTAB: REMOVE TIME STAMP FILES

```
59 * * * * root rm -f /var/spool/cron/lastrun/cron.hourly
14 2 * * * root rm -f /var/spool/cron/lastrun/cron.daily
29 2 * * 6 root rm -f /var/spool/cron/lastrun/cron.weekly
```

```
44 2 1 * *      root  rm -f /var/spool/cron/lastrun/cron.monthly
```

Or you can set `DAILY_TIME` in `/etc/sysconfig/cron` to the time at which `cron.daily` should start. The setting of `MAX_NOT_RUN` ensures that the daily tasks get triggered to run, even if the user did not turn on the computer at the specified `DAILY_TIME` for a longer time. The maximum value of `MAX_NOT_RUN` is 14 days.

The daily system maintenance jobs are distributed to various scripts for reasons of clarity. They are contained in the package `aaa_base`. `/etc/cron.daily` contains, for example, the components `suse.de-backup-rpmdb`, `suse.de-clean-tmp` or `suse.de-cron-local`.

21.1.3 Stopping Cron Status Messages

To avoid the mail-flood caused by cron status messages, the default value of `SEND_MAIL_ON_NO_ERROR` in `/etc/sysconfig/cron` is set to `"no"` for new installations. Even with this setting to `"no"`, cron data output will still be sent to the `MAILTO` address, as documented in the cron man page.

In the update case it is recommended to set these values according to your needs.

21.1.4 Log Files: Package logrotate

There are several system services (*daemons*) that, along with the kernel itself, regularly record the system status and specific events onto log files. This way, the administrator can regularly check the status of the system at a certain point in time, recognize errors or faulty functions and troubleshoot them with pinpoint precision. These log files are normally stored in `/var/log` as specified by FHS and grow on a daily basis. The `logrotate` package helps control the growth of these files.

Configure logrotate with the file

`/etc/logrotate.conf`. In particular, the `include` specification primarily configures the additional files to read. Programs that produce log files install individual configuration files in `/etc/logrotate.d`. For example, such files ship with the packages `apache2` (`/etc/logrotate.d/apache2`) and `syslog-service` (`/etc/logrotate.d/syslog`).

EXAMPLE 21.3: EXAMPLE FOR /ETC/LOGROTATE.CONF

```
# see "man logrotate" for details
```

```
# rotate log files weekly
weekly

# keep 4 weeks worth of backlogs
rotate 4

# create new (empty) log files after rotating old ones
create

# uncomment this if you want your log files compressed
#compress

# RPM packages drop log rotation information into this directory
include /etc/logrotate.d

# no packages own lastlog or wtmp - we'll rotate them here
#/var/log/wtmp {
#   monthly
#   create 0664 root utmp
#   rotate 1
#}

# system-specific logs may be also be configured here.
```

logrotate is controlled through cron and is called daily by /etc/cron.daily/logrotate.

Important: Permissions

The create option reads all settings made by the administrator in /etc/permissions*. Ensure that no conflicts arise from any personal modifications.

21.1.5 The locate Command

locate, a command for quickly finding files, is not included in the standard scope of installed software. If desired, install the package mlocate, the successor of the package findutils-locate. The updatedb process is started automatically every night or about 15 minutes after booting the system.

21.1.6 The ulimit Command

With the **ulimit** (*user limits*) command, it is possible to set limits for the use of system resources and to have these displayed. **ulimit** is especially useful for limiting available memory for applications. With this, an application can be prevented from co-opting too much of the system resources and slowing or even hanging up the operating system.

ulimit can be used with various options. To limit memory usage, use the options listed in *Table 21.1, “ulimit: Setting Resources for the User”*.

TABLE 21.1: ulimit: SETTING RESOURCES FOR THE USER

-m	The maximum resident set size
-v	The maximum amount of virtual memory available to the shell
-s	The maximum size of the stack
-c	The maximum size of core files created
-a	All current limits are reported

Systemwide default entries are set in `/etc/profile`. Editing this file directly is not recommended, because changes will be overwritten during system upgrades. To customize systemwide profile settings, use `/etc/profile.local`. Per-user settings should be made in `~USER/.bashrc`.

EXAMPLE 21.4: ULIMIT: SETTINGS IN ~/.BASHRC

```
# Limits maximum resident set size (physical memory):
ulimit -m 98304

# Limits of virtual memory:
ulimit -v 98304
```

Memory allocations must be specified in KB. For more detailed information, see **man bash**.



Important: ulimit Support

Not all shells support **ulimit** directives. PAM (for instance, `pam_limits`) offers comprehensive adjustment possibilities as an alternative to **ulimit**.

21.1.7 The free Command

The **free** command displays the total amount of free and used physical memory and swap space in the system, plus the buffers and cache consumed by the kernel. The concept of *available RAM* dates back to before the days of unified memory management. The slogan *free memory is bad memory* applies well to Linux. As a result, Linux has always made the effort to balance out caches without actually allowing free or unused memory.

The kernel does not have direct knowledge of any applications or user data. Instead, it manages applications and user data in a *page cache*. If memory runs short, parts of it are written to the swap partition or to files, from which they can initially be read with the help of the **mmap** command (see **man mmap**).

The kernel also contains other caches, such as the *slab cache*, where the caches used for network access are stored. This may explain the differences between the counters in `/proc/meminfo`. Most, but not all, of them can be accessed via `/proc/slabinfo`.

However, if your goal is to find out how much RAM is currently being used, find this information in `/proc/meminfo`.

21.1.8 Man Pages and Info Pages

For some GNU applications (such as tar), the man pages are no longer maintained. For these commands, use the `--help` option to get a quick overview of the info pages, which provide more in-depth instructions. Info is GNU's hypertext system. Read an introduction to this system by entering **info info**. Info pages can be viewed with Emacs by entering **emacs -f info** or directly in a console with **info**. You can also use tinfo, xinfo or the help system to view info pages.

21.1.9 Selecting Man Pages Using the man Command

To read a man page enter **man man_page**. If a man page with the same name exists in different sections, they will all be listed with the corresponding section numbers. Select the one to display. If you do not enter a section number within a few seconds, the first man page will be displayed. If you want to change this to the default system behavior, set `MAN_POSIXLY_CORRECT=1` in a shell initialization file such as `~/.bashrc`.

21.1.10 Settings for GNU Emacs

GNU Emacs is a complex work environment. The following sections cover the configuration files processed when GNU Emacs is started. More information is available at <http://www.gnu.org/software/emacs/>.

On start-up, Emacs reads several files containing the settings of the user, system administrator and distributor for customization or preconfiguration. The initialization file `~/.emacs` is installed to the home directories of the individual users from `/etc/skel/.emacs`, in turn, reads the file `/etc/skel/.gnu-emacs`. To customize the program, copy `.gnu-emacs` to the home directory (with `cp /etc/skel/.gnu-emacs ~/.gnu-emacs`) and make the desired settings there.

`.gnu-emacs` defines the file `~/.gnu-emacs-custom` as `custom-file`. If users make settings with the `customize` options in Emacs, the settings are saved to `~/.gnu-emacs-custom`.

With SUSE Linux Enterprise Server, the `emacs` package installs the file `site-start.el` in the directory `/usr/share/emacs/site-lisp`. The file `site-start.el` is loaded before the initialization file `~/.emacs`. Among other things, `site-start.el` ensures that special configuration files distributed with Emacs add-on packages, such as `psgml`, are loaded automatically. Configuration files of this type are located in `/usr/share/emacs/site-lisp`, too, and always begin with `suse-start-`. The local system administrator can specify systemwide settings in `default.el`.

More information about these files is available in the Emacs info file under *Init File*: `info:/emacs/InitFile`. Information about how to disable the loading of these files (if necessary) is also provided at this location.

The components of Emacs are divided into several packages:

- The base package `emacs`.
- `emacs-x11` (usually installed): the program *with* X11 support.
- `emacs-nox`: the program *without* X11 support.
- `emacs-info`: online documentation in info format.

- emacs-el: the uncompiled library files in Emacs Lisp. These are not required at runtime.
- Numerous add-on packages can be installed if needed: emacs-auctex (LaTeX), psgml (SGML and XML), gnuserv (client and server operation) and others.

21.2 Virtual Consoles

Linux is a multiuser and multitasking system. The advantages of these features can be appreciated even on a stand-alone PC system. In text mode, there are six virtual consoles available. Switch between them using **Alt-F1** through **Alt-F6**. The seventh console is reserved for X and the tenth console shows kernel messages.

To switch to a console from X without shutting it down, use **Ctrl-Alt-F1** to **Ctrl-Alt-F6**. To return to X, press **Alt-F7**.

21.3 Keyboard Mapping

To standardize the keyboard mapping of programs, changes were made to the following files:

```
/etc/inputrc
/etc/X11/Xmodmap
/etc/skel/.emacs
/etc/skel/.gnu-emacs
/etc/skel/.vimrc
/etc/csh.cshrc
/etc/termcap
/usr/share/terminfo/x/xterm
/usr/share/X11/app-defaults/XTerm
/usr/share/emacs/VERSION/site-lisp/term/*.el
```

These changes only affect applications that use terminfo entries or whose configuration files are changed directly (vi, emacs, etc.). Applications not shipped with the system should be adapted to these defaults.

Under X, the compose key (multikey) can be enabled as explained in /etc/X11/Xmodmap.

Further settings are possible using the X Keyboard Extension (XKB). This extension is also used by the desktop environment GNOME (gswitchit).



Tip: For More Information

Information about XKB is available in the documents listed in /usr/share/doc/packages/xkeyboard-config (part of the xkeyboard-config package).

21.4 Language and Country-Specific Settings

The system is, to a very large extent, internationalized and can be modified for local needs. Internationalization (*I18N*) allows specific localization (*L10N*). The abbreviations *I18N* and *L10N* are derived from the first and last letters of the words and, in between, the number of letters omitted.

Settings are made with LC_ variables defined in the file /etc/sysconfig/language. This refers not only to *native language support*, but also to the categories *Messages* (Language), *Character Set*, *Sort Order*, *Time and Date*, *Numbers* and *Money*. Each of these categories can be defined directly with its own variable or indirectly with a master variable in the file language (see the **locale** man page).

RC_LC_MESSAGES, RC_LC_CTYPE, RC_LC_COLLATE, RC_LC_TIME, RC_LC_NUMERIC,
RC_LC_MONETARY

These variables are passed to the shell without the RC_ prefix and represent the listed categories. The shell profiles concerned are listed below. The current setting can be shown with the command **locale**.

RC_LC_ALL

This variable, if set, overwrites the values of the variables already mentioned.

RC_LANG

If none of the previous variables are set, this is the fallback. By default, only RC_LANG is set. This makes it easier for users to enter their own values.

ROOT_USES_LANG

A yes or no variable. If set to no, root always works in the POSIX environment.

The variables can be set with the YaST sysconfig editor. The value of such a variable contains the language code, country code, encoding and modifier. The individual components are connected by special characters:

```
LANG=<language>[[<COUNTRY>].<Encoding>[@<Modifier>]]
```

21.4.1 Some Examples

You should always set the language and country codes together. Language settings follow the standard ISO 639 available at <http://www.evertype.com/standards/iso639/iso639-en.html> and <http://www.loc.gov/standards/iso639-2/>. Country codes are listed in ISO 3166, see http://en.wikipedia.org/wiki/ISO_3166.

It only makes sense to set values for which usable description files can be found in `/usr/lib/locale`. Additional description files can be created from the files in `/usr/share/i18n` using the command `localedef`. The description files are part of the `glibc-i18ndata` package. A description file for `en_US.UTF-8` (for English and United States) can be created with:

```
localedef -i en_US -f UTF-8 en_US.UTF-8
```

LANG=en_US.UTF-8

This is the default setting if American English is selected during installation. If you selected another language, that language is enabled but still with UTF-8 as the character encoding.

LANG=en_US.ISO-8859-1

This sets the language to English, country to United States and the character set to ISO-8859-1. This character set does not support the Euro sign, but it can be useful sometimes for programs that have not been updated to support UTF-8. The string defining the charset (ISO-8859-1 in this case) is then evaluated by programs like Emacs.

LANG=en_IE@euro

The above example explicitly includes the Euro sign in a language setting. This setting is obsolete now, as UTF-8 also covers the Euro symbol. It is only useful if an application supports ISO-8859-15 and not UTF-8.

Changes to /etc/sysconfig/language are activated by the following process chain:

- For the Bash: /etc/profile reads /etc/profile.d/lang.sh which, in turn, analyzes /etc/sysconfig/language.
- For tcsh: At login, /etc/csh.login reads /etc/profile.d/lang.csh which, in turn, analyzes /etc/sysconfig/language.

This ensures that any changes to `/etc/sysconfig/language` are available at the next login to the respective shell, without having to manually activate them.

Users can override the system defaults by editing their `~/.bashrc` accordingly. For instance, if you do not want to use the system-wide `en_US` for program messages, include `LC_MESSAGES=es_ES` so that messages are displayed in Spanish instead.

21.4.2 Locale Settings in `~/.i18n`

If you are not satisfied with locale system defaults, change the settings in `~/.i18n` according to the Bash scripting syntax. Entries in `~/.i18n` override system defaults from `/etc/sysconfig/language`. Use the same variable names but without the `RC_` name space prefixes. For example, use `LANG` instead of `RC_LANG`:

```
LANG=cs_CZ.UTF-8
LC_COLLATE=C
```

21.4.3 Settings for Language Support

Files in the category *Messages* are, as a rule, only stored in the corresponding language directory (like `en`) to have a fallback. If you set `LANG` to `en_US` and the message file in `/usr/share/locale/en_US/LC_MESSAGES` does not exist, it falls back to `/usr/share/locale/en/LC_MESSAGES`.

A fallback chain can also be defined, for example, for Breton to French or for Galician to Spanish to Portuguese:

```
LANGUAGE="br_FR:fr_FR"
```

```
LANGUAGE="gl_ES:es_ES:pt_PT"
```

If desired, use the Norwegian variants Nynorsk and Bokmål instead (with additional fallback to `no`):

```
LANG="nn_NO"
```

```
LANGUAGE="nn_NO:nb_NO:no"
```

or

```
LANG="nb_NO"
```

```
LANGUAGE="nb_NO:nn_NO:no"
```

Note that in Norwegian, `LC_TIME` is also treated differently.

One problem that can arise is a separator used to delimit groups of digits not being recognized properly. This occurs if `LANG` is set to only a two-letter language code like `de`, but the definition file glibc uses is located in `/usr/share/lib/de_DE/LC_NUMERIC`. Thus `LC_NUMERIC` must be set to `de_DE` to make the separator definition visible to the system.

21.4.4 For More Information

- *The GNU C Library Reference Manual*, Chapter “Locales and Internationalization”. It is included in `glibc-info`. The package is available from the SUSE Linux Enterprise SDK. The SDK is a module for SUSE Linux Enterprise and is available via an online channel from the SUSE Customer Center. Alternatively, go to <http://download.suse.com/>, search for SUSE Linux Enterprise Software Development Kit and download it from there. Refer to *Book “Deployment Guide”, Chapter 13 “Installing Modules, Extensions, and Third Party Add-On Products”* for details.
- Markus Kuhn, *UTF-8 and Unicode FAQ for Unix/Linux*, currently at <http://www.cl.cam.ac.uk/~mgk25/unicode.html>.
- *Unicode-HOWTO* by Bruno Haible, available at <http://tldp.org/HOWTO/Unicode-HOWTO-1.html>.

III Services

- 22 Time Synchronization with NTP **318**
- 23 The Domain Name System **325**
- 24 DHCP **350**
- 25 Sharing File Systems with NFS **366**
- 26 Samba **377**
- 27 On-Demand Mounting with Autofs **399**
- 28 SLP **407**
- 29 The Apache HTTP Server **411**
- 30 Setting Up an FTP Server with YaST **454**
- 31 The Proxy Server Squid **458**
- 32 Web Based Enterprise Management Using SFCB **481**

22 Time Synchronization with NTP

The NTP (network time protocol) mechanism is a protocol for synchronizing the system time over the network. First, a machine can obtain the time from a server that is a reliable time source. Second, a machine can itself act as a time source for other computers in the network. The goal is twofold—maintaining the absolute time and synchronizing the system time of all machines within a network.

Maintaining an exact system time is important in many situations. The built-in hardware clock does often not meet the requirements of applications such as databases or clusters. Manual correction of the system time would lead to severe problems because, for example, a backward leap can cause malfunction of critical applications. Within a network, it is usually necessary to synchronize the system time of all machines, but manual time adjustment is a bad approach. NTP provides a mechanism to solve these problems. The NTP service continuously adjusts the system time with reliable time servers in the network. It further enables the management of local reference clocks, such as radio-controlled clocks.

22.1 Configuring an NTP Client with YaST

The NTP daemon (`ntpd`) coming with the `ntp` package is preset to use the local computer clock as a time reference. Using the hardware clock, however, only serves as a fallback for cases where no time source of better precision is available. YaST simplifies the configuration of an NTP client.

22.1.1 Basic Configuration

The YaST NTP client configuration (*Network Services > NTP Configuration*) consists of tabs. Set the start mode of `ntpd` and the server to query on the *General Settings* tab.

Only Manually

Select *Only Manually*, if you want to manually start the `ntpd` daemon.

Synchronize without Daemon

Select *Synchronize without Daemon* to set the system time periodically without a permanently running `ntpd`. You can set the *Interval of the Synchronization in Minutes*.

Now and On Boot

Select *Now and On Boot* to start `ntpd` automatically when the system is booted. This setting is recommended.

22.1.2 Changing Basic Configuration

The servers and other time sources for the client to query are listed in the lower part of the *General Settings* tab. Modify this list as needed with *Add*, *Edit*, and *Delete*. *Display Log* provides the possibility to view the log files of your client.

Click *Add* to add a new source of time information. In the following dialog, select the type of source with which the time synchronization should be made. The following options are available:

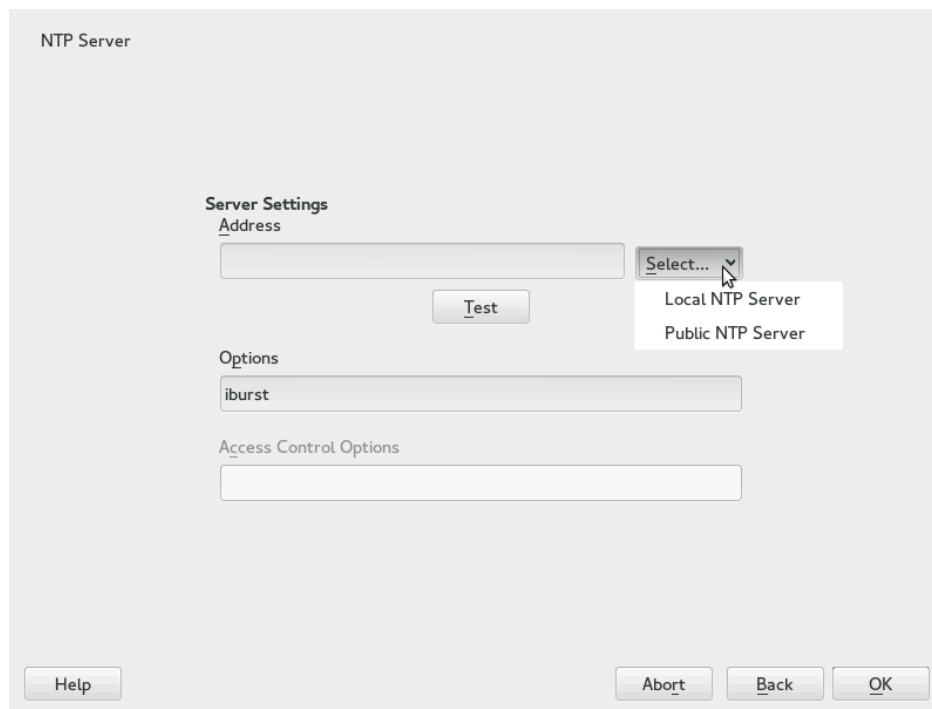


FIGURE 22.1: YAST: NTP SERVER

Server

In the pull-down *Select* list (see *Figure 22.1, “YaST: NTP Server”*), determine whether to set up time synchronization using a time server from your local network (*Local NTP Server*) or an Internet-based time server that takes care of your time zone (*Public NTP Server*). For a local time server, click *Lookup* to start an SLP query for available time servers in your

network. Select the most suitable time server from the list of search results and exit the dialog with *OK*. For a public time server, select your country (time zone) and a suitable server from the list under *Public NTP Server* then exit the dialog with *OK*. In the main dialog, test the availability of the selected server with *Test*. *Options* allows you to specify additional options for `ntpd`.

Using *Access Control Options*, you can restrict the actions that the remote computer can perform with the daemon running on your computer. This field is enabled only after checking *Restrict NTP Service to Configured Servers Only* on the *Security Settings* tab (see [Figure 22.2, "Advanced NTP Configuration: Security Settings"](#)). The options correspond to the `restrict` clauses in `/etc/ntp.conf`. For example, `nomodify notrap noquery` disallows the server to modify NTP settings of your computer and to use the trap facility (a remote event logging feature) of your NTP daemon. Using these restrictions is recommended for servers out of your control (for example, on the Internet).

Refer to `/usr/share/doc/packages/ntp-doc` (part of the `ntp-doc` package) for detailed information.

Peer

A peer is a machine to which a symmetric relationship is established: it acts both as a time server and as a client. To use a peer in the same network instead of a server, enter the address of the system. The rest of the dialog is identical to the *Server* dialog.

Radio Clock

To use a radio clock in your system for the time synchronization, enter the clock type, unit number, device name, and other options in this dialog. Click *Driver Calibration* to fine-tune the driver. Detailed information about the operation of a local radio clock is available in [/usr/share/doc/packages/ntp-doc/refclock.html](#).

Outgoing Broadcast

Time information and queries can also be transmitted by broadcast in the network. In this dialog, enter the address to which such broadcasts should be sent. Do not activate broadcasting unless you have a reliable time source like a radio controlled clock.

Incoming Broadcast

If you want your client to receive its information via broadcast, enter the address from which the respective packets should be accepted in this fields.

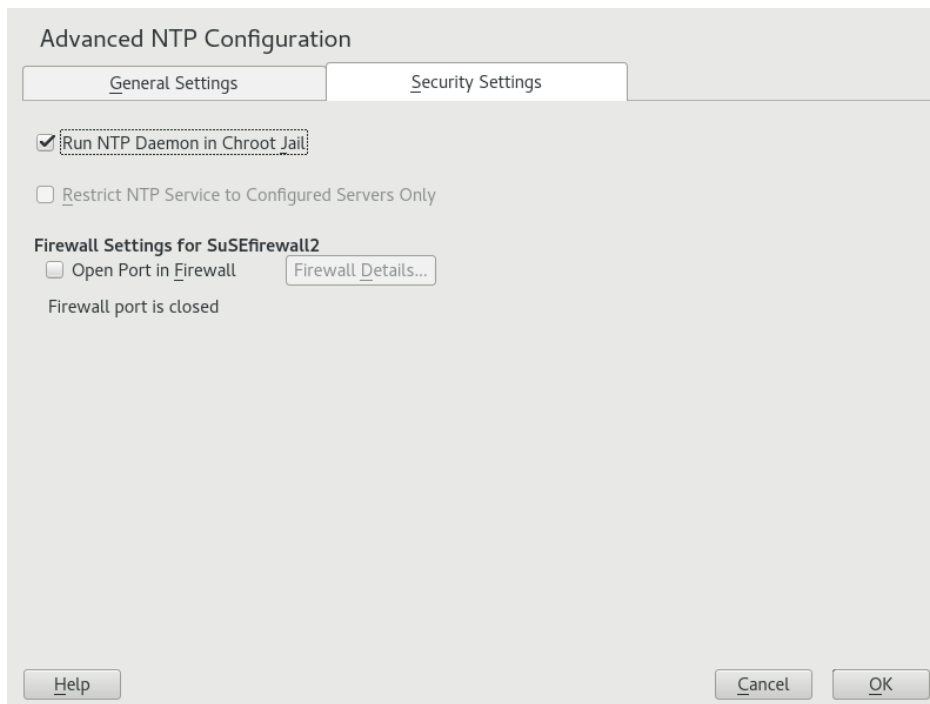


FIGURE 22.2: ADVANCED NTP CONFIGURATION: SECURITY SETTINGS

In the *Security Settings* tab (see [Figure 22.2, “Advanced NTP Configuration: Security Settings”](#)), determine whether `ntpd` should be started in a chroot jail. By default, *Run NTP Daemon in Chroot Jail* is not activated. The chroot jail option increases the security in the event of an attack over `ntpd`, as it prevents the attacker from compromising the entire system.

Restrict NTP Service to Configured Servers Only increases the security of your system by disallowing remote computers to view and modify NTP settings of your computer and to use the trap facility for remote event logging. After being enabled, these restrictions apply to all remote computers, unless you override the access control options for individual computers in the list of time sources in the *General Settings* tab. For all other remote computers, only querying for local time is allowed.

Enable *Open Port in Firewall* if SuSEFirewall2 is active (which it is by default). If you leave the port closed, it is not possible to establish a connection to the time server.

22.2 Manually Configuring NTP in the Network

The easiest way to use a time server in the network is to set server parameters. For example, if a time server called `ntp.example.com` is reachable from the network, add its name to the file `/etc/ntp.conf` by adding the following line:

```
server ntp.example.com
```

To add more time servers, insert additional lines with the keyword `server`. After initializing `ntpd` with the command `systemctl start ntp`, it takes about one hour until the time is stabilized and the drift file for correcting the local computer clock is created. With the drift file, the systematic error of the hardware clock can be computed when the computer is powered on. The correction is used immediately, resulting in a higher stability of the system time.

There are two possible ways to use the NTP mechanism as a client: First, the client can query the time from a known server in regular intervals. With many clients, this approach can cause a high load on the server. Second, the client can wait for NTP broadcasts sent out by broadcast time servers in the network. This approach has the disadvantage that the quality of the server is unknown and a server sending out wrong information can cause severe problems.

If the time is obtained via broadcast, you do not need the server name. In this case, enter the line `broadcastclient` in the configuration file `/etc/ntp.conf`. To use one or more known time servers exclusively, enter their names in the line starting with `servers`.

22.3 Dynamic Time Synchronization at Runtime

If the system boots without network connection, `ntpd` starts up, but it cannot resolve DNS names of the time servers set in the configuration file. This can happen if you use NetworkManager with an encrypted Wi-Fi.

If you want `ntpd` to resolve DNS names at runtime, you must set the `dynamic` option. Then, when the network is established some time after booting, `ntpd` looks up the names again and can reach the time servers to get the time.

Manually edit `/etc/ntp.conf` and add `dynamic` to one or more `server` entries:

```
server ntp.example.com dynamic
```

Or use YaST and proceed as follows:

1. In YaST click *Network Services* > *NTP Configuration*.
2. Select the server you want to configure. Then click *Edit*.
3. Activate the *Options* field and add dynamic. Separate it with a space, if there are already other options entered.
4. Click *Ok* to close the edit dialog. Repeat the previous step to change all servers as wanted.
5. Finally click *Ok* to save the settings.

22.4 Setting Up a Local Reference Clock

The software package ntpd contains drivers for connecting local reference clocks. A list of supported clocks is available in the ntp-doc package in the file /usr/share/doc/packages/ntp-doc/refclock.html. Every driver is associated with a number. In NTP, the actual configuration takes place by means of pseudo IP addresses. The clocks are entered in the file /etc/ntp.conf as though they existed in the network. For this purpose, they are assigned special IP addresses in the form 127.127.t.u. Here, t stands for the type of the clock and determines which driver is used and u for the unit, which determines the interface used.

Normally, the individual drivers have special parameters that describe configuration details. The file /usr/share/doc/packages/ntp-doc/drivers/driverNN.html (where NN is the number of the driver) provides information about the particular type of clock. For example, the “type 8” clock (radio clock over serial interface) requires an additional mode that specifies the clock more precisely. The Conrad DCF77 receiver module, for example, has mode 5. To use this clock as a preferred reference, specify the keyword prefer. The complete server line for a Conrad DCF77 receiver module would be:

```
server 127.127.8.0 mode 5 prefer
```

Other clocks follow the same pattern. Following the installation of the ntp-doc package, the documentation for NTP is available in the directory /usr/share/doc/packages/ntp-doc. The file /usr/share/doc/packages/ntp-doc/refclock.html provides links to the driver pages describing the driver parameters.

22.5 Clock Synchronization to an External Time Reference (ETR)

Support for clock synchronization to an external time reference (ETR) is available. The external time reference sends an oscillator signal and a synchronization signal every 2^{20} (2 to the power of 20) microseconds to keep TOD clocks of all connected servers synchronized.

For availability two ETR units can be connected to a machine. If the clock deviates for more than the sync-check tolerance all CPUs get a machine check that indicates that the clock is out of sync. If this happens, all DASD I/O to XRC enabled devices is stopped until the clock is synchronized again.

The ETR support is activated via two `sysfs` attributes; run the following commands as `root`:

```
echo 1 > /sys/devices/system/etr/etr0/online
echo 1 > /sys/devices/system/etr/etr1/online
```

23 The Domain Name System

DNS (domain name system) is needed to resolve the domain names and host names into IP addresses. In this way, the IP address 192.168.2.100 is assigned to the host name jupiter, for example. Before setting up your own name server, read the general information about DNS in *Section 15.3, “Name Resolution”*. The following configuration examples refer to BIND, the default DNS server.

23.1 DNS Terminology

Zone

The domain name space is divided into regions called zones. For instance, if you have example.com, you have the example section (or zone) of the com domain.

DNS server

The DNS server is a server that maintains the name and IP information for a domain. You can have a primary DNS server for master zone, a secondary server for slave zone, or a slave server without any zones for caching.

Master zone DNS server

The master zone includes all hosts from your network and a DNS server master zone stores up-to-date records for all the hosts in your domain.

Slave zone DNS server

A slave zone is a copy of the master zone. The slave zone DNS server obtains its zone data with zone transfer operations from its master server. The slave zone DNS server responds authoritatively for the zone as long as it has valid (not expired) zone data. If the slave cannot obtain a new copy of the zone data, it stops responding for the zone.

Forwarder

Forwarders are DNS servers to which your DNS server should send queries it cannot answer. To enable different configuration sources in one configuration, netconfig is used (see also man 8 netconfig).

Record

The record is information about name and IP address. Supported records and their syntax are described in BIND documentation. Some special records are:

NS record

An NS record tells name servers which machines are in charge of a given domain zone.

MX record

The MX (mail exchange) records describe the machines to contact for directing mail across the Internet.

SOA record

SOA (Start of Authority) record is the first record in a zone file. The SOA record is used when using DNS to synchronize data between multiple computers.

23.2 Installation

To install a DNS server, start YaST and select *Software > Software Management*. Choose *View > Patterns* and select *DHCP and DNS Server*. Confirm the installation of the dependent packages to finish the installation process.

23.3 Configuration with YaST

Use the YaST DNS module to configure a DNS server for the local network. When starting the module for the first time, a wizard starts, prompting you to make a few decisions concerning administration of the server. Completing this initial setup produces a basic server configuration. Use the expert mode to deal with more advanced configuration tasks, such as setting up ACLs, logging, TSIG keys, and other options.

23.3.1 Wizard Configuration

The wizard consists of three steps or dialogs. At the appropriate places in the dialogs, you can enter the expert configuration mode.

1. When starting the module for the first time, the *Forwarder Settings* dialog, shown in *Figure 23.1, “DNS Server Installation: Forwarder Settings”*, opens. The *Local DNS Resolution Policy* allows to set the following options:

- *Merging forwarders is disabled*
- *Automatic merging*
- *Merging forwarders is enabled*
- *Custom configuration*—If *Custom configuration* is selected, *Custom policy* can be specified; by default (with *Automatic merging* selected), *Custom policy* is set to auto, but here you can either set interface names or select from the two special policy names STATIC and STATIC_FALLBACK.

In *Local DNS Resolution Forwarder*, specify which service to use: *Using system name servers*, *This name server (bind)*, or *Local dnsmasq server*.

For more information about all these settings, see man 8 netconfig.

DNS Server Installation: Forwarder Settings

Local DNS Resolution Policy: Automatic merging

Custom policy: auto

Local DNS Resolution Forwarder: This name server (bind)

Add IP Address

IPv4 or IPv6 Address: 192.168.27.1

Add

Forwarder List

192.168.27.1

Delete

Help Cancel Back Next

FIGURE 23.1: DNS SERVER INSTALLATION: FORWARDER SETTINGS

Forwarders are DNS servers to which your DNS server sends queries it cannot answer itself. Enter their IP address and click *Add*.



23.3.2 Expert Configuration

After starting the module, YaST opens a window displaying several configuration options. Completing it results in a DNS server configuration with the basic functions in place:

23.3.2.1 Start-Up

Under *Start-Up*, define whether the DNS server should be started when booting the system or manually. To start the DNS server immediately, click *Start DNS Server Now*. To stop the DNS server, click *Stop DNS Server Now*. To save the current settings, select *Save Settings and Reload DNS Server Now*. You can open the DNS port in the firewall with *Open Port in Firewall* and modify the firewall settings with *Firewall Details*.

By selecting *LDAP Support Active*, the zone files are managed by an LDAP database. Any changes to zone data written to the LDAP database are picked up by the DNS server when it is restarted or prompted to reload its configuration.

23.3.2.2 Forwarders

If your local DNS server cannot answer a request, it tries to forward the request to a *Forwarder*, if configured so. This forwarder may be added manually to the *Forwarder List*. If the forwarder is not static like in dial-up connections, *netconfig* handles the configuration. For more information about *netconfig*, see [man 8 netconfig](#).

23.3.2.3 Basic Options

In this section, set basic server options. From the *Option* menu, select the desired item then specify the value in the corresponding text box. Include the new entry by selecting *Add*.

23.3.2.4 Logging

To set what the DNS server should log and how, select *Logging*. Under *Log Type*, specify where the DNS server should write the log data. Use the system-wide log by selecting *System Log* or specify a different file by selecting *File*. In the latter case, additionally specify a name, the maximum file size in megabytes and the number of log file versions to store.

Further options are available under *Additional Logging*. Enabling *Log All DNS Queries* causes *every* query to be logged, in which case the log file could grow extremely large. For this reason, it is not a good idea to enable this option for other than debugging purposes. To log the data traffic during zone updates between DHCP and DNS server, enable *Log Zone Updates*. To log the data traffic during a zone transfer from master to slave, enable *Log Zone Transfer*. See [*Figure 23.4, "DNS Server: Logging"*](#).

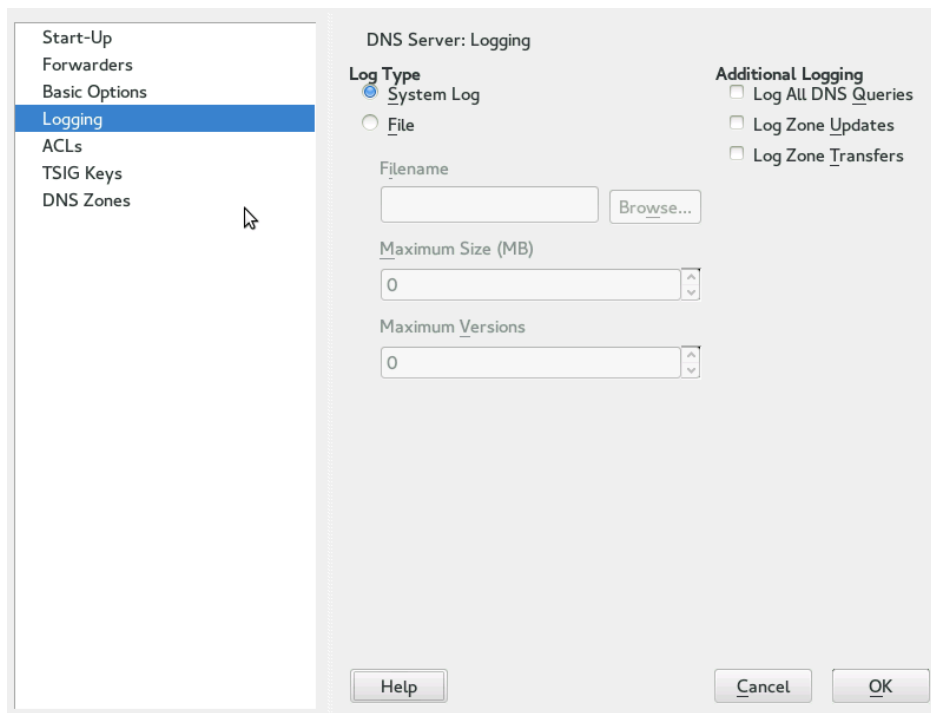


FIGURE 23.4: DNS SERVER: LOGGING

23.3.2.5 ACLs

Use this dialog to define ACLs (access control lists) to enforce access restrictions. After providing a distinct name under *Name*, specify an IP address (with or without netmask) under *Value* in the following fashion:

```
{ 192.168.1/24; }
```

The syntax of the configuration file requires that the address ends with a semicolon and is put into curly braces.

23.3.2.6 TSIG Keys

The main purpose of TSIGs (transaction signatures) is to secure communications between DHCP and DNS servers. They are described in [Section 23.8, “Secure Transactions”](#).

To generate a TSIG key, enter a distinctive name in the field labeled *Key ID* and specify the file where the key should be stored (*Filename*). Confirm your choices with *Generate*.

To use a previously created key, leave the *Key ID* field blank and select the file where it is stored under *Filename*. After that, confirm with *Add*.

23.3.2.7 DNS Zones (Adding a Slave Zone)

To add a slave zone, select *DNS Zones*, choose the zone type *Slave*, write the name of the new zone, and click *Add*.

In the *Zone Editor* sub-dialog under *Master DNS Server IP*, specify the master from which the slave should pull its data. To limit access to the server, select one of the ACLs from the list.

23.3.2.8 DNS Zones (Adding a Master Zone)

To add a master zone, select *DNS Zones*, choose the zone type *Master*, write the name of the new zone, and click *Add*. When adding a master zone, a reverse zone is also needed. For example, when adding the zone example.com that points to hosts in a subnet 192.168.1.0/24, you should also add a reverse zone for the IP-address range covered. By definition, this should be named 1.168.192.in-addr.arpa.

23.3.2.9 DNS Zones (Editing a Master Zone)

To edit a master zone, select *DNS Zones*, select the master zone from the table, and click *Edit*. The dialog consists of several pages: *Basics* (the one opened first), *NS Records*, *MX Records*, *SOA*, and *Records*.

The basic dialog, shown in *Figure 23.5, "DNS Server: Zone Editor (Basics)"*, lets you define settings for dynamic DNS and access options for zone transfers to clients and slave name servers. To permit the dynamic updating of zones, select *Allow Dynamic Updates* as well as the corresponding TSIG key. The key must have been defined before the update action starts. To enable zone transfers, select the corresponding ACLs. ACLs must have been defined already.

In the *Basics* dialog, select whether to enable zone transfers. Use the listed ACLs to define who can download zones.

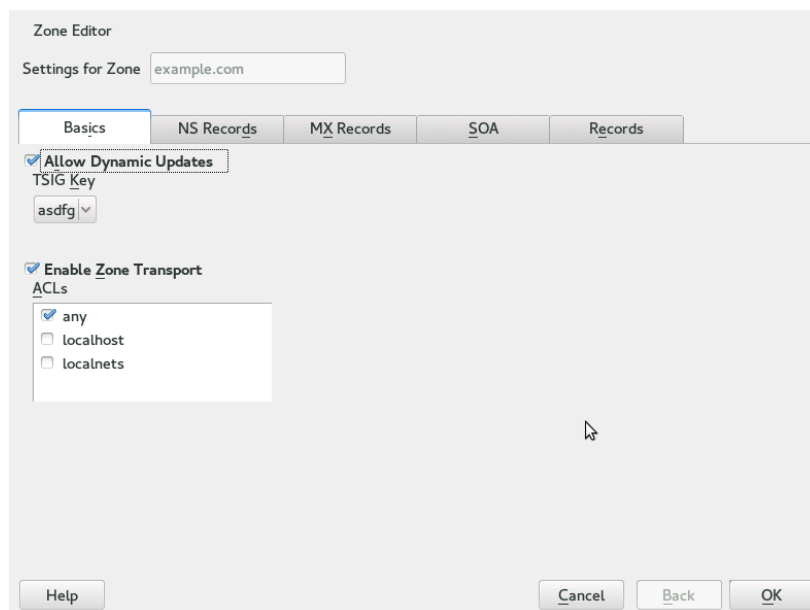


FIGURE 23.5: DNS SERVER: ZONE EDITOR (BASICS)

Zone Editor (NS Records)

The *NS Records* dialog allows you to define alternative name servers for the zones specified. Make sure that your own name server is included in the list. To add a record, enter its name under *Name Server to Add* then confirm with *Add*. See *Figure 23.6, "DNS Server: Zone Editor (NS Records)"*.

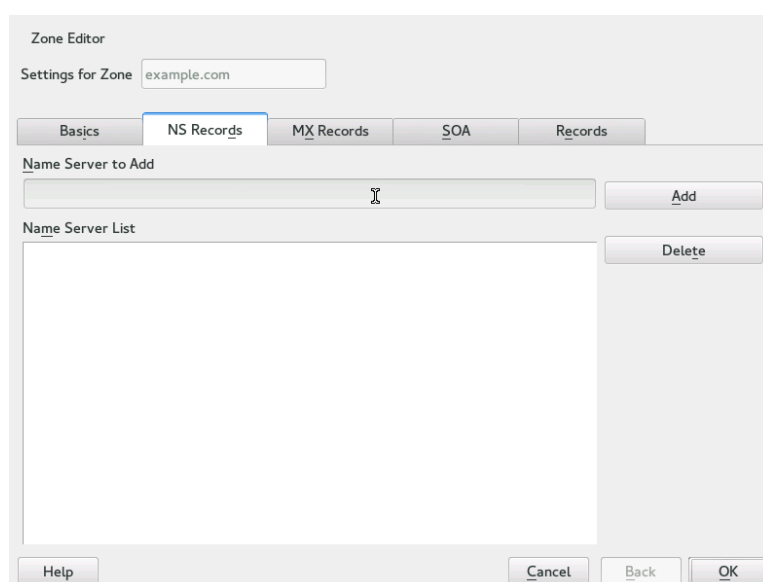


FIGURE 23.6: DNS SERVER: ZONE EDITOR (NS RECORDS)

Zone Editor (MX Records)

To add a mail server for the current zone to the existing list, enter the corresponding address and priority value. After doing so, confirm by selecting *Add*. See *Figure 23.7, “DNS Server: Zone Editor (MX Records)”*.

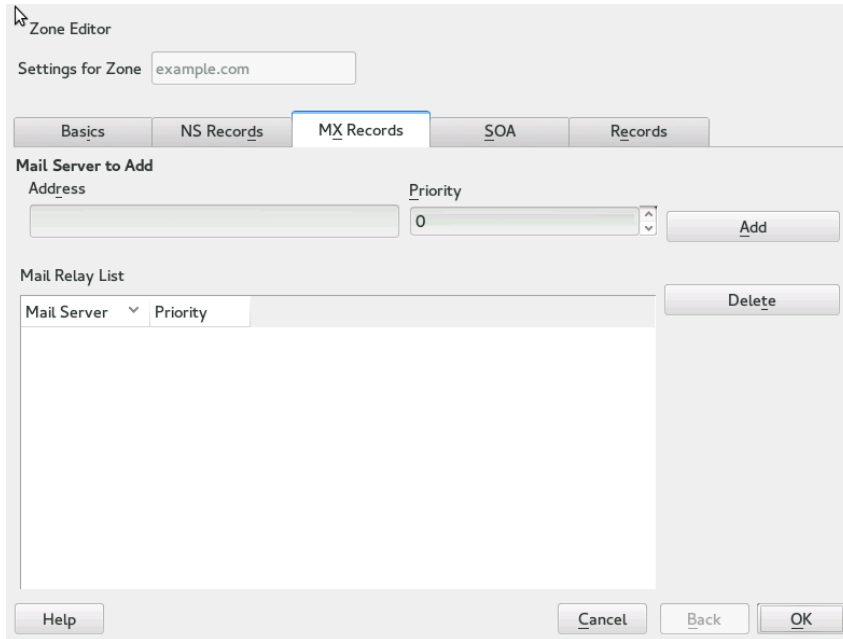


FIGURE 23.7: DNS SERVER: ZONE EDITOR (MX RECORDS)

Zone Editor (SOA)

This page allows you to create SOA (start of authority) records. For an explanation of the individual options, refer to *Example 23.6, “The /var/lib/named/example.com.zone File”*. Changing SOA records is not supported for dynamic zones managed via LDAP.

Zone Editor

Settings for Zone

Basics NS Records MX Records **SOA** Records

Serial

TTL Unit

Refresh Unit

Retry Unit

Expiration Unit

Minimum Unit

Help Cancel Back OK

FIGURE 23.8: DNS SERVER: ZONE EDITOR (SOA)

Zone Editor (Records)

This dialog manages name resolution. In *Record Key*, enter the host name then select its type. The *A* type represents the main entry. The value for this should be an IP address (IPv4). Use *AAAA* for IPv6 addresses. *CNAME* is an alias. Use the types *NS* and *MX* for detailed or partial records that expand on the information provided in the *NS Records* and *MX Records* tabs. These three types resolve to an existing *A* record. *PTR* is for reverse zones. It is the opposite of an *A* record, for example:

```
hostname.example.com. IN A 192.168.0.1
1.0.168.192.in-addr.arpa IN PTR hostname.example.com.
```

23.3.2.9.1 Adding Reverse Zones

To add a reverse zone, follow this procedure:

1. Start *YaST* > *DNS Server* > *DNS Zones*.
2. If you have not added a master forward zone, add it and *Edit* it.

3. In the *Records* tab, fill the corresponding *Record Key* and *Value*, then add the record with *Add* and confirm with *OK*. If YaST complains about a non-existing record for a name server, add it in the *NS Records* tab.

The screenshot shows the 'Zone Editor' window for the zone 'example.com'. The 'Records' tab is selected. In the 'Record Settings' section, the 'Record Key' is 'example.com.', the 'Type' is 'A: IPv4 Domain Name Translator', and the 'Value' is '1.1.1.1'. There are 'Change' and 'Add' buttons. Below, the 'Configured Resource Records' table shows one record: 'example.com.' of type 'A' with value '1.1.1.1'. A 'Delete' button is next to the table. At the bottom are 'Help', 'Cancel', 'Back', and 'OK' buttons.

Record Key	Type	Value
example.com.	A	1.1.1.1

FIGURE 23.9: ADDING A RECORD FOR A MASTER ZONE

4. Back in the *DNS Zones* window, add a reverse master zone.

The screenshot shows the 'DNS Server: DNS Zones' window. On the left is a sidebar with a tree view containing 'Start-Up', 'Forwarders', 'Basic Options', 'Logging', 'ACLs', 'TSIG Keys', and 'DNS Zones' (selected). The main area has a checkbox for 'LDAP Support Active' (unchecked). Below is the 'Add New Zone' section with 'Name' set to '1.1.1.1.in-addr.arpa' and 'Type' set to 'Master'. There is an 'Add' button. Below that is the 'Configured DNS Zones' table with two entries: '1.1.1.1.in-addr.arpa' (Master) and 'example.com' (Master). There are 'Delete' and 'Edit' buttons next to the table. At the bottom are 'Help', 'Cancel', and 'OK' buttons.

Zone	Type
1.1.1.1.in-addr.arpa	Master
example.com	Master

FIGURE 23.10: ADDING A REVERSE ZONE

5. *Edit* the reverse zone, and in the *Records* tab, you can see the *PTR: Reverse translation* record type. Add the corresponding *Record Key* and *Value*, then click *Add* and confirm with *OK*.

Zone Editor

Settings for Zone

Basics NS Records SOA **Records**

Record Settings

Record Key	Type	Value
<input type="text" value="1.1.1.1.in-addr.arpa."/>	<input type="text" value="PTR: Reverse Translat"/>	<input type="text" value="example.com."/>

Configured Resource Records

Record Key	Type	Value
1.1.1.1.in-addr.arpa.	PTR	example.com.

FIGURE 23.11: ADDING A REVERSE RECORD

Add a name server record if needed.



Tip: Editing the Reverse Zone

After adding a forward zone, go back to the main menu and select the reverse zone for editing. There in the tab *Basics* activate the check box *Automatically Generate Records From* and select your forward zone. That way, all changes to the forward zone are automatically updated in the reverse zone.

23.4 Starting the BIND Name Server

On a SUSE® Linux Enterprise Server system, the name server BIND (*Berkeley Internet Name Domain*) comes preconfigured so it can be started right after installation without any problems. If you already have a functioning Internet connection and have entered `127.0.0.1` as the name server address for `localhost` in `/etc/resolv.conf`, you normally already have a working name resolution without needing to know the DNS of the provider. BIND carries out name resolution via the root name server, a notably slower process. Normally, the DNS of the provider should be entered with its IP address in the configuration file `/etc/named.conf` under `for-`

warders to ensure effective and secure name resolution. If this works so far, the name server runs as a pure *caching-only* name server. Only when you configure its own zones it becomes a proper DNS. Find a simple example documented in /usr/share/doc/packages/bind/config.



Tip: Automatic Adaptation of the Name Server Information

Depending on the type of Internet connection or the network connection, the name server information can automatically be adapted to the current conditions. To do this, set the NETCONFIG_DNS_POLICY variable in the /etc/sysconfig/network/config file to auto.

However, do not set up an official domain until one is assigned to you by the responsible institution. Even if you have your own domain and it is managed by the provider, you are better off not using it, because BIND would otherwise not forward requests for this domain. The Web server at the provider, for example, would not be accessible for this domain.

To start the name server, enter the command **systemctl start named** as root. Check with **systemctl status named** whether named (as the name server process is called) has been started successfully. Test the name server immediately on the local system with the host or dig programs, which should return localhost as the default server with the address 127.0.0.1. If this is not the case, /etc/resolv.conf probably contains an incorrect name server entry or the file does not exist. For the first test, enter host 127.0.0.1, which should always work. If you get an error message, use **systemctl status named** to see whether the server is actually running. If the name server does not start or behaves unexpectedly, check the output of **journalctl -e**.

To use the name server of the provider (or one already running on your network) as the forwarder, enter the corresponding IP address or addresses in the options section under forwarders. The addresses included in *Example 23.1, "Forwarding Options in named.conf"* are examples only. Adjust these entries to your own setup.

EXAMPLE 23.1: FORWARDING OPTIONS IN NAMED.CONF

```
options {
    directory "/var/lib/named";
    forwarders { 10.11.12.13; 10.11.12.14; };
    listen-on { 127.0.0.1; 192.168.1.116; };
    allow-query { 127/8; 192.168/16 };
    notify no;
```

```
};
```

The `options` entry is followed by entries for the zone, `localhost`, and `0.0.127.in-addr.arpa`. The `type hint` entry under “.” should always be present. The corresponding files do not need to be modified and should work as they are. Also make sure that each entry is closed with a “;” and that the curly braces are in the correct places. After changing the configuration file `/etc/named.conf` or the zone files, tell BIND to reread them with `systemctl reload named`. Achieve the same by stopping and restarting the name server with `systemctl restart named`. Stop the server at any time by entering `systemctl stop named`.

23.5 The `/etc/named.conf` Configuration File

All the settings for the BIND name server itself are stored in the `/etc/named.conf` file. However, the zone data for the domains to handle (consisting of the host names, IP addresses, and so on) are stored in separate files in the `/var/lib/named` directory. The details of this are described later.

`/etc/named.conf` is roughly divided into two areas. One is the `options` section for general settings and the other consists of `zone` entries for the individual domains. A `logging` section and `acl` (access control list) entries are optional. Comment lines begin with a `#` sign or `//`. A minimal `/etc/named.conf` is shown in *Example 23.2, “A Basic `/etc/named.conf`”*.

EXAMPLE 23.2: A BASIC `/ETC/NAMED.CONF`

```
options {
    directory "/var/lib/named";
    forwarders { 10.0.0.1; };
    notify no;
};

zone "localhost" in {
    type master;
    file "localhost.zone";
};

zone "0.0.127.in-addr.arpa" in {
    type master;
    file "127.0.0.zone";
};
```

```
zone "." in {  
    type hint;  
    file "root.hint";  
};
```

23.5.1 Important Configuration Options

directory " filename";

Specifies the directory in which BIND can find the files containing the zone data. Usually, this is /var/lib/named.

forwarders { ip-address;};

Specifies the name servers (mostly of the provider) to which DNS requests should be forwarded if they cannot be resolved directly. Replace ip-address with an IP address like 192.168.1.116.

forward first;

Causes DNS requests to be forwarded before an attempt is made to resolve them via the root name servers. Instead of forward first, forward only can be written to have all requests forwarded and none sent to the root name servers. This makes sense for firewall configurations.

listen-on port 53 { 127.0.0.1; ip-address;};

Tells BIND on which network interfaces and port to accept client queries. port 53 does not need to be specified explicitly, because 53 is the default port. Enter 127.0.0.1 to permit requests from the local host. If you omit this entry entirely, all interfaces are used by default.

listen-on-v6 port 53 {any;};

Tells BIND on which port it should listen for IPv6 client requests. The only alternative to any is none. As far as IPv6 is concerned, the server only accepts wild card addresses.

query-source address * port 53;

This entry is necessary if a firewall is blocking outgoing DNS requests. This tells BIND to post requests externally from port 53 and not from any of the high ports above 1024.

query-source-v6 address * port 53;

Tells BIND which port to use for IPv6 queries.

allow-query { 127.0.0.1; net ;};

Defines the networks from which clients can post DNS requests. Replace net with address information like 192.168.2.0/24. The /24 at the end is an abbreviated expression for the netmask (in this case 255.255.255.0).

allow-transfer ! *;;

Controls which hosts can request zone transfers. In the example, such requests are completely denied with ! *. Without this entry, zone transfers can be requested from anywhere without restrictions.

statistics-interval 0;

In the absence of this entry, BIND generates several lines of statistical information per hour in the system's journal. Set it to 0 to suppress these statistics completely or set an interval in minutes.

cleaning-interval 720;

This option defines at which time intervals BIND clears its cache. This triggers an entry in the system's journal each time it occurs. The time specification is in minutes. The default is 60 minutes.

interface-interval 0;

BIND regularly searches the network interfaces for new or nonexistent interfaces. If this value is set to 0, this is not done and BIND only listens at the interfaces detected at start-up. Otherwise, the interval can be defined in minutes. The default is sixty minutes.

notify no;

no prevents other name servers from being informed when changes are made to the zone data or when the name server is restarted.

For a list of available options, read the manual page man 5 named.conf.

23.5.2 Logging

What, how, and where logging takes place can be extensively configured in BIND. Normally, the default settings should be sufficient. *Example 23.3, "Entry to Disable Logging"*, shows the simplest form of such an entry and completely suppresses any logging.

EXAMPLE 23.3: ENTRY TO DISABLE LOGGING

```
logging {
```

```
category default { null; };  
};
```

23.5.3 Zone Entries

EXAMPLE 23.4: ZONE ENTRY FOR EXAMPLE.COM

```
zone "example.com" in {  
    type master;  
    file "example.com.zone";  
    notify no;  
};
```

After zone, specify the name of the domain to administer (example.com) followed by in and a block of relevant options enclosed in curly braces, as shown in *Example 23.4, "Zone Entry for example.com"*. To define a *slave zone*, switch the type to slave and specify a name server that administers this zone as master (which, in turn, may be a slave of another master), as shown in *Example 23.5, "Zone Entry for example.net"*.

EXAMPLE 23.5: ZONE ENTRY FOR EXAMPLE.NET

```
zone "example.net" in {  
    type slave;  
    file "slave/example.net.zone";  
    masters { 10.0.0.1; };  
};
```

The zone options:

type master;

By specifying master, tell BIND that the zone is handled by the local name server. This assumes that a zone file has been created in the correct format.

type slave;

This zone is transferred from another name server. It must be used together with masters.

type hint;

The zone . of the hint type is used to set the root name servers. This zone definition can be left as is.

file example.com.zone or file "slave/example.net.zone";

This entry specifies the file where zone data for the domain is located. This file is not required for a slave, because this data is pulled from another name server. To differentiate master and slave files, use the directory slave for the slave files.

masters { server-ip-address;;

This entry is only needed for slave zones. It specifies from which name server the zone file should be transferred.

allow-update {! *};

This option controls external write access, which would allow clients to make a DNS entry—something not normally desirable for security reasons. Without this entry, zone updates are not allowed. The above entry achieves the same because ! * effectively bans any such activity.

23.6 Zone Files

Two types of zone files are needed. One assigns IP addresses to host names and the other does the reverse: it supplies a host name for an IP address.



Tip: Using the Dot (Period, Fullstop) in Zone Files

The "." has an important meaning in the zone files. If host names are given without a final dot (.), the zone is appended. Complete host names specified with a full domain name must end with a dot (.) to avoid having the domain added to it again. A missing or wrongly placed "." is probably the most frequent cause of name server configuration errors.

The first case to consider is the zone file example.com.zone, responsible for the domain example.com, shown in *Example 23.6, "The /var/lib/named/example.com.zone File"*.

EXAMPLE 23.6: THE /VAR/LIB/NAMED/EXAMPLE.COM.ZONE FILE

```
1. $TTL 2D
2. example.com. IN SOA      dns root.example.com. (
3.                2003072441 ; serial
4.                1D        ; refresh
```



```

5.          2H          ; retry
6.          1W          ; expiry
7.          2D )        ; minimum
8.
9.          IN NS       dns
10.         IN MX       10 mail
11.
12. gate     IN A        192.168.5.1
13.         IN A        10.0.0.1
14. dns      IN A        192.168.1.116
15. mail     IN A        192.168.3.108
16. jupiter  IN A        192.168.2.100
17. venus    IN A        192.168.2.101
18. saturn   IN A        192.168.2.102
19. mercury  IN A        192.168.2.103
20. ntp      IN CNAME    dns
21. dns6     IN A6       0 2002:c0a8:174::

```

Line 1:

\$TTL defines the default time to live that should apply to all the entries in this file. In this example, entries are valid for a period of two days (2 D).

Line 2:

This is where the SOA (start of authority) control record begins:

- The name of the domain to administer is example.com in the first position. This ends with ".", because otherwise the zone would be appended a second time. Alternatively, @ can be entered here, in which case the zone would be extracted from the corresponding entry in /etc/named.conf.
- After IN SOA is the name of the name server in charge as master for this zone. The name is expanded from dns to dns.example.com, because it does not end with a ".".
- An e-mail address of the person in charge of this name server follows. Because the @ sign already has a special meaning, "." is entered here instead. For root@example.com the entry must read root.example.com.. The "." must be included at the end to prevent the zone from being added.
- The (includes all lines up to) into the SOA record.

Line 3:

The serial number is an arbitrary number that is increased each time this file is changed. It is needed to inform the secondary name servers (slave servers) of changes. For this, a 10 digit number of the date and run number, written as YYYYMMDDNN, has become the customary format.

Line 4:

The refresh rate specifies the time interval at which the secondary name servers verify the zone serial number. In this case, one day.

Line 5:

The retry rate specifies the time interval at which a secondary name server, in case of error, attempts to contact the primary server again. Here, two hours.

Line 6:

The expiration time specifies the time frame after which a secondary name server discards the cached data if it has not regained contact to the primary server. Here, a week.

Line 7:

The last entry in the SOA record specifies the negative caching TTL—the time for which results of unresolved DNS queries from other servers may be cached.

Line 9:

The IN NS specifies the name server responsible for this domain. dns is extended to dns.example.com because it does not end with a ".". There can be several lines like this—one for the primary and one for each secondary name server. If notify is not set to no in /etc/named.conf, all the name servers listed here are informed of the changes made to the zone data.

Line 10:

The MX record specifies the mail server that accepts, processes, and forwards e-mails for the domain example.com. In this example, this is the host mail.example.com. The number in front of the host name is the preference value. If there are multiple MX entries, the mail server with the smallest value is taken first and, if mail delivery to this server fails, an attempt is made with the next higher value.

Lines 12–19:

These are the actual address records where one or more IP addresses are assigned to host names. The names are listed here without a `"."` because they do not include their domain, so `example.com` is added to all of them. Two IP addresses are assigned to the host `gate`, as it has two network cards. Wherever the host address is a traditional one (IPv4), the record is marked with `A`. If the address is an IPv6 address, the entry is marked with `AAAA`.



Note: IPv6 Syntax

The IPv6 record has a slightly different syntax than IPv4. Because of the fragmentation possibility, it is necessary to provide information about missed bits before the address. To fill up the IPv6 address with the needed number of “0”, add two colons at the correct place in the address.

```
pluto      AAAA 2345:00C1:CA11::1234:5678:9ABC:DEF0
pluto      AAAA 2345:00D2:DA11::1234:5678:9ABC:DEF0
```

Line 20:

The alias `ntp` can be used to address `dns` (`CNAME` means *canonical name*).

The pseudo domain `in-addr.arpa` is used for the reverse lookup of IP addresses into host names. It is appended to the network part of the address in reverse notation. So `192.168` is resolved into `168.192.in-addr.arpa`. See [Example 23.7, “Reverse Lookup”](#).

EXAMPLE 23.7: REVERSE LOOKUP

```
1. $TTL 2D
2. 168.192.in-addr.arpa.  IN SOA dns.example.com. root.example.com. (
3.                        2003072441      ; serial
4.                        1D              ; refresh
5.                        2H              ; retry
6.                        1W              ; expiry
7.                        2D )            ; minimum
8.
9.                        IN NS          dns.example.com.
10.
11. 1.5                      IN PTR    gate.example.com.
12. 100.3                    IN PTR    www.example.com.
13. 253.2                    IN PTR    cups.example.com.
```

Line 1:

\$TTL defines the standard TTL that applies to all entries here.

Line 2:

The configuration file should activate reverse lookup for the network 192.168.. Given that the zone is called 168.192.in-addr.arpa, it should not be added to the host names. Therefore, all host names are entered in their complete form—with their domain and with a "." at the end. The remaining entries correspond to those described for the previous example.com example.

Lines 3–7:

See the previous example for example.com.

Line 9:

Again this line specifies the name server responsible for this zone. This time, however, the name is entered in its complete form with the domain and a "." at the end.

Lines 11–13:

These are the pointer records hinting at the IP addresses on the respective hosts. Only the last part of the IP address is entered at the beginning of the line, without the "." at the end. Appending the zone to this (without the .in-addr.arpa) results in the complete IP address in reverse order.

Normally, zone transfers between different versions of BIND should be possible without any problems.

23.7 Dynamic Update of Zone Data

The term *dynamic update* refers to operations by which entries in the zone files of a master server are added, changed, or deleted. This mechanism is described in RFC 2136. Dynamic update is configured individually for each zone entry by adding an optional allow-update or update-policy rule. Zones to update dynamically should not be edited by hand.

Transmit the entries to update to the server with the command **nsupdate**. For the exact syntax of this command, check the manual page for nsupdate (**man 8 nsupdate**). For security reasons, any such update should be performed using TSIG keys as described in [Section 23.8, "Secure Transactions"](#).

23.8 Secure Transactions

Secure transactions can be made with transaction signatures (TSIGs) based on shared secret keys (also called TSIG keys). This section describes how to generate and use such keys.

Secure transactions are needed for communication between different servers and for the dynamic update of zone data. Making the access control dependent on keys is much more secure than merely relying on IP addresses.

Generate a TSIG key with the following command (for details, see [`man dnssec-keygen`](#)):

```
dnssec-keygen -a hmac-md5 -b 128 -n HOST host1-host2
```

This creates two files with names similar to these:

```
Khost1-host2.+157+34265.private Khost1-host2.+157+34265.key
```

The key itself (a string like `ejIkuCyyGJwwuN3xAteKgg==`) is found in both files. To use it for transactions, the second file (`Khost1-host2.+157+34265.key`) must be transferred to the remote host, preferably in a secure way (using `scp`, for example). On the remote server, the key must be included in the `/etc/named.conf` file to enable a secure communication between `host1` and `host2`:

```
key host1-host2 {  
    algorithm hmac-md5;  
    secret "ejIkuCyyGJwwuN3xAteKgg=";  
};
```



Warning: File Permissions of `/etc/named.conf`

Make sure that the permissions of `/etc/named.conf` are properly restricted. The default for this file is `0640`, with the owner being `root` and the group `named`. As an alternative, move the keys to an extra file with specially limited permissions, which is then included from `/etc/named.conf`. To include an external file, use:

```
include "filename"
```

Replace `filename` with an absolute path to your file with keys.

To enable the server host1 to use the key for host2 (which has the address 10.1.2.3 in this example), the server's /etc/named.conf must include the following rule:

```
server 10.1.2.3 {  
    keys { host1-host2. ;};  
};
```

Analogous entries must be included in the configuration files of host2.

Add TSIG keys for any ACLs (access control lists, not to be confused with file system ACLs) that are defined for IP addresses and address ranges to enable transaction security. The corresponding entry could look like this:

```
allow-update { key host1-host2. ;};
```

This topic is discussed in more detail in the *BIND Administrator Reference Manual* under update-policy.

23.9 DNS Security

DNSSEC, or DNS security, is described in RFC 2535. The tools available for DNSSEC are discussed in the BIND Manual.

A zone considered secure must have one or several zone keys associated with it. These are generated with dnssec-keygen, as are the host keys. The DSA encryption algorithm is currently used to generate these keys. The public keys generated should be included in the corresponding zone file with an \$INCLUDE rule.

With the command dnssec-signzone, you can create sets of generated keys (keyset - files), transfer them to the parent zone in a secure manner, and sign them. This generates the files to include for each zone in /etc/named.conf.

23.10 For More Information

For more information, see the *BIND Administrator Reference Manual* from the bind-doc package, which is installed under /usr/share/doc/packages/bind/arm. Consider additionally consulting the RFCs referenced by the manual and the manual pages included with BIND. /usr/share/doc/packages/bind/README.SUSE contains up-to-date information about BIND in SUSE Linux Enterprise Server.

24 DHCP

The purpose of the *Dynamic Host Configuration Protocol* (DHCP) is to assign network settings centrally (from a server) rather than configuring them locally on every workstation. A host configured to use DHCP does not have control over its own static address. It is enabled to configure itself completely and automatically according to directions from the server. If you use the NetworkManager on the client side, you do not need to configure the client. This is useful if you have changing environments and only one interface active at a time. Never use NetworkManager on a machine that runs a DHCP server.



Tip: IBM z Systems: DHCP Support

On IBM z Systems platforms, DHCP only works on interfaces using the OSA and OSA Express network cards. These cards are the only ones with a MAC, which is required for the DHCP autoconfiguration features.

One way to configure a DHCP server is to identify each client using the hardware address of its network card (which should be fixed in most cases), then supply that client with identical settings each time it connects to the server. DHCP can also be configured to assign addresses to each relevant client dynamically from an address pool set up for this purpose. In the latter case, the DHCP server tries to assign the same address to the client each time it receives a request, even over extended periods. This works only if the network does not have more clients than addresses.

DHCP makes life easier for system administrators. Any changes, even bigger ones, related to addresses and the network configuration in general can be implemented centrally by editing the server's configuration file. This is much more convenient than reconfiguring numerous workstations. It is also much easier to integrate machines, particularly new machines, into the network, because they can be given an IP address from the pool. Retrieving the appropriate network settings from a DHCP server is especially useful in case of laptops regularly used in different networks.

In this chapter, the DHCP server will run in the same subnet as the workstations, 192.168.2.0/24 with 192.168.2.1 as gateway. It has the fixed IP address 192.168.2.254 and serves two address ranges, 192.168.2.10 to 192.168.2.20 and 192.168.2.100 to 192.168.2.200.

A DHCP server supplies not only the IP address and the netmask, but also the host name, domain name, gateway, and name server addresses for the client to use. In addition to that, DHCP allows several other parameters to be configured in a centralized way, for example, a time server from which clients may poll the current time or even a print server.

24.1 Configuring a DHCP Server with YaST

To install a DHCP server, start YaST and select *Software > Software Management*. Choose *Filter > Patterns* and select *DHCP and DNS Server*. Confirm the installation of the dependent packages to finish the installation process.



Important: LDAP Support

The YaST DHCP module can be set up to store the server configuration locally (on the host that runs the DHCP server) or to have its configuration data managed by an LDAP server. If you want to use LDAP, set up your LDAP environment before configuring the DHCP server.

For more information about LDAP, see *Book “Security Guide”, Chapter 5 “LDAP—A Directory Service”*.

The YaST DHCP module (`yast2-dhcp-server`) allows you to set up your own DHCP server for the local network. The module can run in wizard mode or expert configuration mode.

24.1.1 Initial Configuration (Wizard)

When the module is started for the first time, a wizard starts, prompting you to make a few basic decisions concerning server administration. Completing this initial setup produces a very basic server configuration that should function in its essential aspects. The expert mode can be used to deal with more advanced configuration tasks. Proceed as follows:

1. Select the interface from the list to which the DHCP server should listen and click *Select*. After this, select *Open Firewall for Selected Interfaces* to open the firewall for this interface, and click *Next*. See *Figure 24.1, “DHCP Server: Card Selection”*.

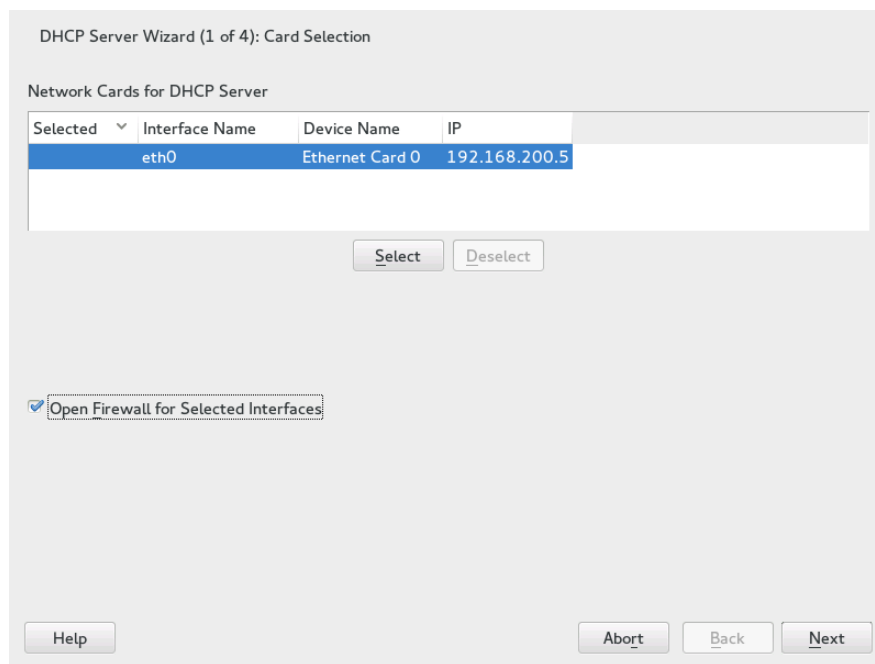


FIGURE 24.1: DHCP SERVER: CARD SELECTION

2. Use the check box to determine whether your DHCP settings should be automatically stored by an LDAP server. In the text boxes, provide the network specifics for all clients the DHCP server should manage. These specifics are the domain name, address of a time server, addresses of the primary and secondary name server, addresses of a print and a WINS server (for a mixed network with both Windows and Linux clients), gateway address, and lease time. See *Figure 24.2, "DHCP Server: Global Settings"*.

DHCP Server Wizard (2 of 4): Global Settings

☐ LDAP Support

DHCP Server Name (optional)

Domain Name: example.org

Primary Name Server IP: 192.168.200.2

Secondary Name Server IP: 192.168.200.3

Default Gateway (Router): 192.168.200.1

NTP Time Server: 192.168.200.10

Print Server:

WINS Server:

Default Lease Time: 1

Units: Hours

Help Abort Back Next

FIGURE 24.2: DHCP SERVER: GLOBAL SETTINGS

3. Configure how dynamic IP addresses should be assigned to clients. To do so, specify an IP range from which the server can assign addresses to DHCP clients. All these addresses must be covered by the same netmask. Also specify the lease time during which a client may keep its IP address without needing to request an extension of the lease. Optionally, specify the maximum lease time—the period during which the server reserves an IP address for a particular client. See *Figure 24.3, “DHCP Server: Dynamic DHCP”*.

DHCP Server Wizard (3 of 4): Dynamic DHCP

Subnet Information

Current Network	Current Netmask	Netmask Bits
<input type="text" value="192.168.200.0"/>	<input type="text" value="255.255.255.0"/>	<input type="text" value="24"/>
Minimum IP Address	Maximum IP Address	
<input type="text" value="192.168.200.1"/>	<input type="text" value="192.168.200.254"/>	

IP Address Range

First IP Address	Last IP Address
<input type="text" value="192.168.200.11"/>	<input type="text" value="192.168.200.254"/>

☐ Allow Dynamic BOOTP

Lease Time

Default	Units	Maximum	Units
<input type="text" value="1"/>	<input type="text" value="Hours"/>	<input type="text" value="2"/>	<input type="text" value="Days"/>

FIGURE 24.3: DHCP SERVER: DYNAMIC DHCP

4. Define how the DHCP server should be started. Specify whether to start the DHCP server automatically when the system is booted or manually when needed (for example, for testing purposes). Click *Finish* to complete the configuration of the server. See [Figure 24.4, "DHCP Server: Start-Up"](#).



FIGURE 24.4: DHCP SERVER: START-UP

5. Instead of using dynamic DHCP in the way described in the preceding steps, you can also configure the server to assign addresses in quasi-static fashion. Use the text boxes provided in the lower part to specify a list of the clients to manage in this way. Specifically, provide the *Name* and the *IP Address* to give to such a client, the *Hardware Address*, and the *Network Type* (token ring or Ethernet). Modify the list of clients, which is shown in the upper part with *Add*, *Edit*, and *Delete from List*. See [Figure 24.5, "DHCP Server: Host Management"](#).

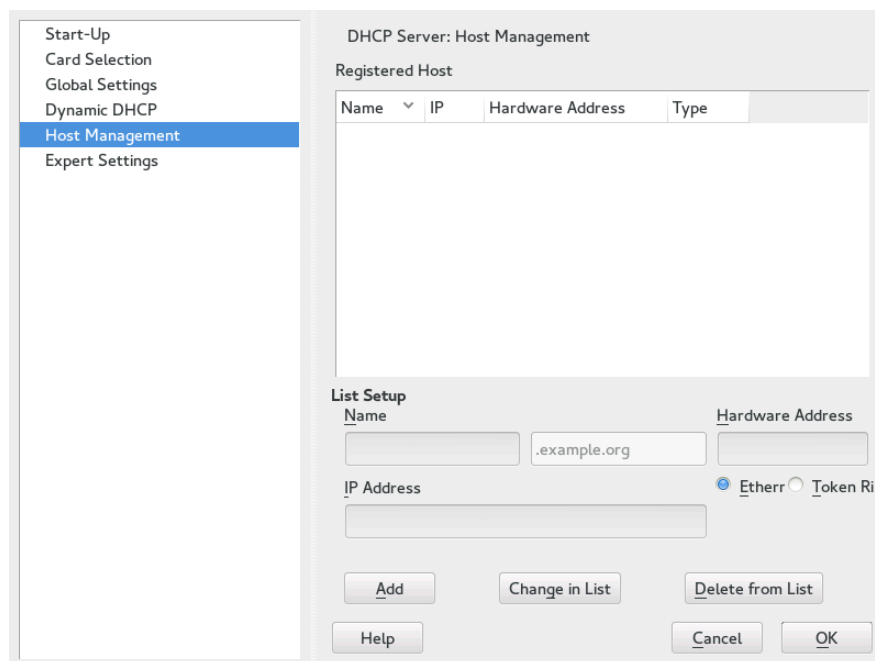


FIGURE 24.5: DHCP SERVER: HOST MANAGEMENT

24.1.2 DHCP Server Configuration (Expert)

In addition to the configuration method discussed earlier, there is also an expert configuration mode that allows you to change the DHCP server setup in every detail. Start the expert configuration by clicking *DHCP Server Expert Configuration* in the *Start-Up* dialog (see [Figure 24.4](#), “*DHCP Server: Start-Up*”).

Chroot Environment and Declarations

In this first dialog, make the existing configuration editable by selecting *Start DHCP Server*. An important feature of the behavior of the DHCP server is its ability to run in a chroot environment or chroot jail, to secure the server host. If the DHCP server should ever be compromised by an outside attack, the attacker will still be behind bars in the chroot jail, which prevents him from touching the rest of the system. The lower part of the dialog displays a tree view with the declarations that have already been defined. Modify these with *Add*, *Delete*, and *Edit*. Selecting *Advanced* takes you to additional expert dialogs. See [Figure 24.6](#), “*DHCP Server: Chroot Jail and Declarations*”. After selecting *Add*, define the type of declaration to add. With *Advanced*, view the log file of the server, configure TSIG key management, and adjust the configuration of the firewall according to the setup of the DHCP server.

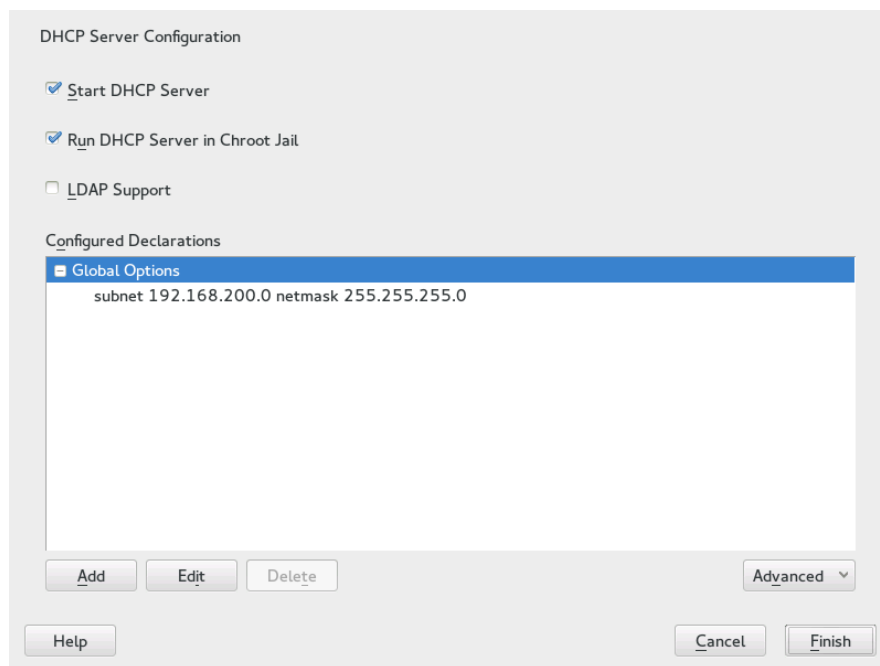


FIGURE 24.6: DHCP SERVER: CHROOT JAIL AND DECLARATIONS

Selecting the Declaration Type

The *Global Options* of the DHCP server are made up of several declarations. This dialog lets you set the declaration types *Subnet*, *Host*, *Shared Network*, *Group*, *Pool of Addresses*, and *Class*. This example shows the selection of a new subnet (see [Figure 24.7, “DHCP Server: Selecting a Declaration Type”](#)).

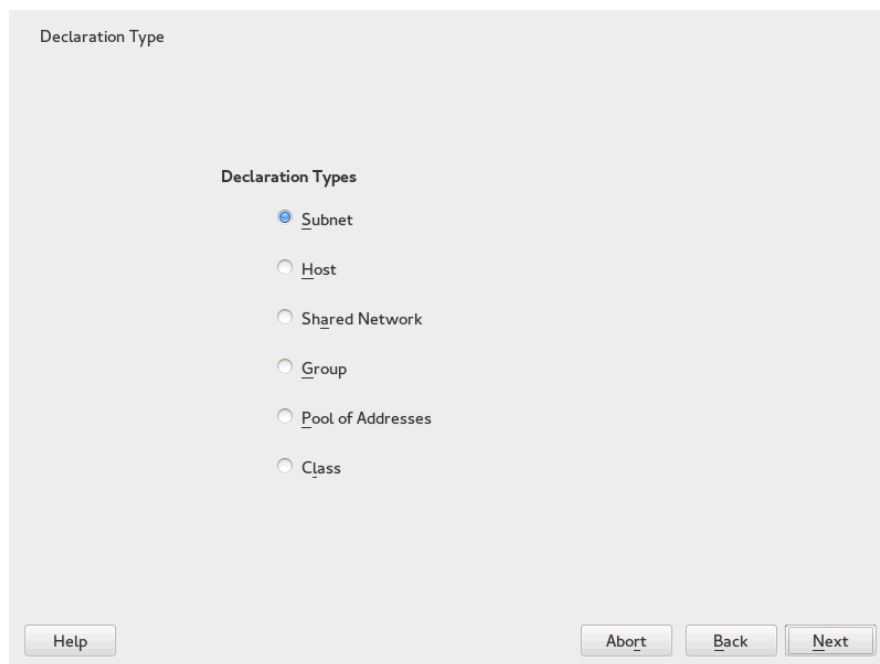


FIGURE 24.7: DHCP SERVER: SELECTING A DECLARATION TYPE

Subnet Configuration

This dialog allows you specify a new subnet with its IP address and netmask. In the middle part of the dialog, modify the DHCP server start options for the selected subnet using *Add*, *Edit*, and *Delete*. To set up dynamic DNS for the subnet, select *Dynamic DNS*.

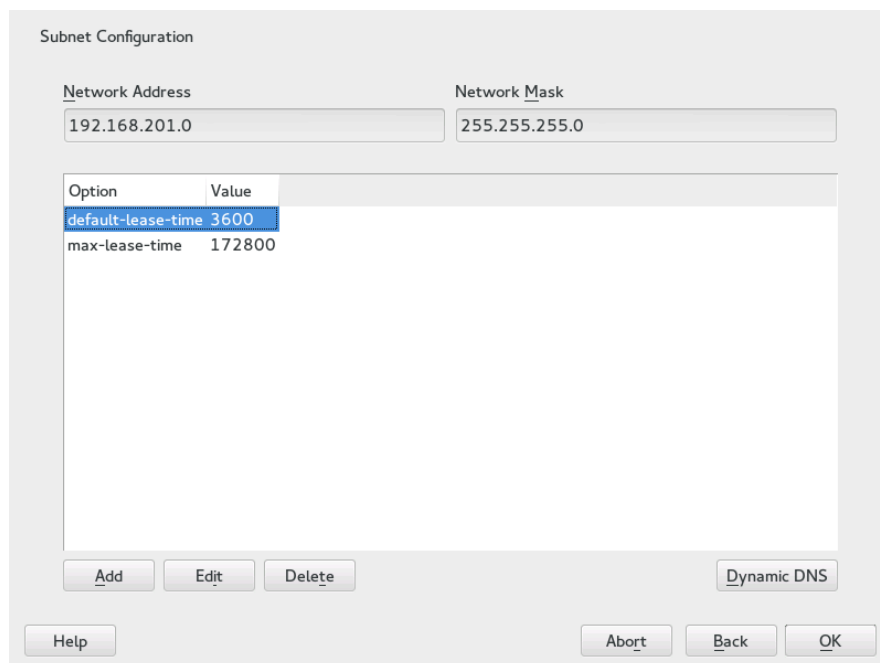


FIGURE 24.8: DHCP SERVER: CONFIGURING SUBNETS

TSIG Key Management

If you chose to configure dynamic DNS in the previous dialog, you can now configure the key management for a secure zone transfer. Selecting *OK* takes you to another dialog in which to configure the interface for dynamic DNS (see [Figure 24.10, “DHCP Server: Interface Configuration for Dynamic DNS”](#)).

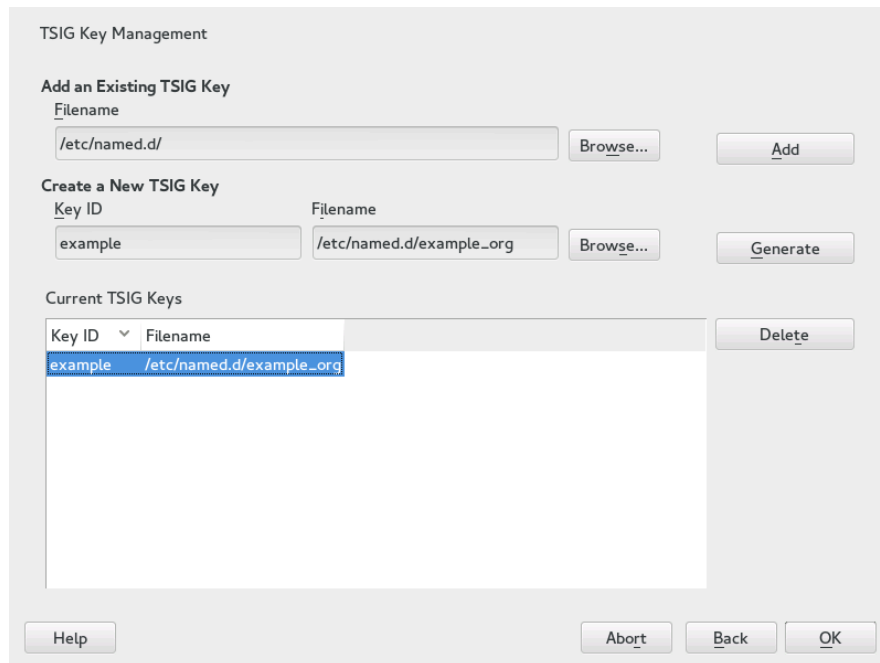


FIGURE 24.9: DHCP SERVER: TSIG CONFIGURATION

Dynamic DNS: Interface Configuration

You can now activate dynamic DNS for the subnet by selecting *Enable Dynamic DNS for This Subnet*. After doing so, use the drop-down box to activate the TSIG keys for forward and reverse zones, making sure that the keys are the same for the DNS and the DHCP server. With *Update Global Dynamic DNS Settings*, enable the automatic update and adjustment of the global DHCP server settings according to the dynamic DNS environment. Finally, define which forward and reverse zones should be updated per dynamic DNS, specifying the name of the primary name server for each of the two zones. Selecting *OK* returns to the subnet configuration dialog (see [Figure 24.8, “DHCP Server: Configuring Subnets”](#)). Selecting *OK* again returns to the original expert configuration dialog.

Interface Configuration

☒ Enable Dynamic DNS for This Subnet

Forward Zone TSIG Key
example

Reverse Zone TSIG Key
example

☐ Update Global Dynamic DNS Settings

Zone
Primary DNS Server

Reverse Zone
Primary DNS Server

Help Abort Back OK

FIGURE 24.10: DHCP SERVER: INTERFACE CONFIGURATION FOR DYNAMIC DNS

Network Interface Configuration

To define the interfaces the DHCP server should listen to and to adjust the firewall configuration, select *Advanced > Interface Configuration* from the expert configuration dialog. From the list of interfaces displayed, select one or more that should be attended by the DHCP server. If clients in all subnets need to be able to communicate with the server and the server host also runs a firewall, adjust the firewall accordingly. To do so, select *Adapt Firewall Settings*. YaST then adjusts the rules of SuSEFirewall2 to the new conditions (see *Figure 24.11, “DHCP Server: Network Interface and Firewall”*), after which you can return to the original dialog by selecting *OK*.

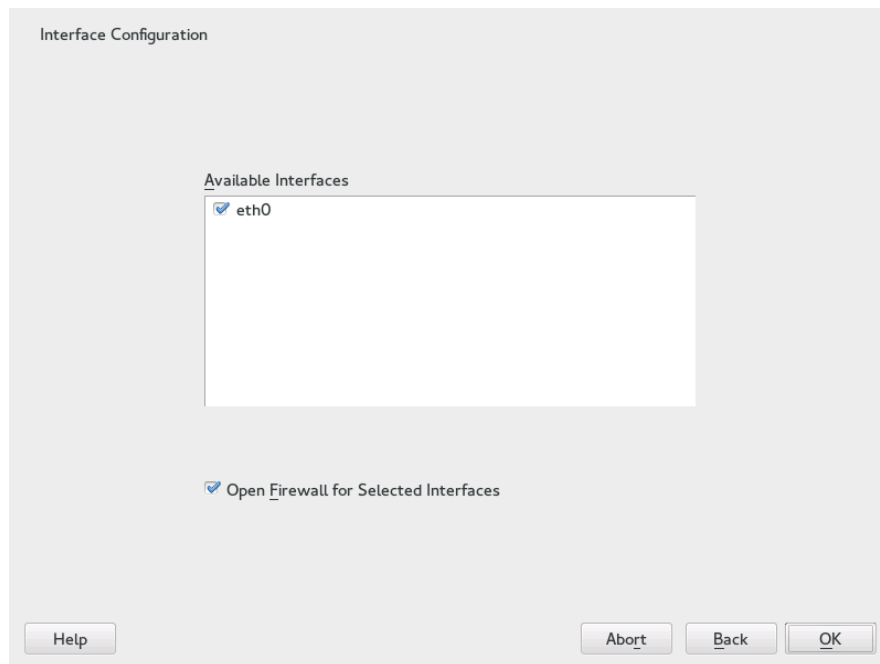


FIGURE 24.11: DHCP SERVER: NETWORK INTERFACE AND FIREWALL

After completing all configuration steps, close the dialog with *OK*. The server is now started with its new configuration.

24.2 DHCP Software Packages

Both the DHCP server and the DHCP clients are available for SUSE Linux Enterprise Server. The DHCP server available is dhcpcd (published by the Internet Systems Consortium). On the client side, there is dhcp-client (also from ISC) and tools coming with the wicked package.

By default, the wicked tools are installed with the services wickedd-dhcp4 and wickedd-dhcp6. Both are launched automatically on each system boot to watch for a DHCP server. They do not need a configuration file to do their job and work out of the box in most standard setups. For more complex situations, use the ISC dhcp-client, which is controlled by means of the configuration files /etc/dhclient.conf and /etc/dhclient6.conf.

24.3 The DHCP Server dhcpd

The core of any DHCP system is the dynamic host configuration protocol daemon. This server *leases* addresses and watches how they are used, according to the settings defined in the configuration file /etc/dhcpd.conf. By changing the parameters and values in this file, a system administrator can influence the program's behavior in numerous ways. Look at the basic sample /etc/dhcpd.conf file in *Example 24.1, "The Configuration File /etc/dhcpd.conf"*.

EXAMPLE 24.1: THE CONFIGURATION FILE /ETC/DHCPD.CONF

```
default-lease-time 600;          # 10 minutes
max-lease-time 7200;             # 2  hours

option domain-name "example.com";
option domain-name-servers 192.168.1.116;
option broadcast-address 192.168.2.255;
option routers 192.168.2.1;
option subnet-mask 255.255.255.0;

subnet 192.168.2.0 netmask 255.255.255.0
{
    range 192.168.2.10 192.168.2.20;
    range 192.168.2.100 192.168.2.200;
}
```

This simple configuration file should be sufficient to get the DHCP server to assign IP addresses in the network. Make sure that a semicolon is inserted at the end of each line, because otherwise `dhcpd` is not started.

The sample file can be divided into three sections. The first one defines how many seconds an IP address is leased to a requesting client by default (default-lease-time) before it should apply for renewal. This section also includes a statement of the maximum period for which a machine may keep an IP address assigned by the DHCP server without applying for renewal (max-lease-time).

In the second part, some basic network parameters are defined on a global level:

- The line option domain-name defines the default domain of your network.
- With the entry option domain-name-servers, specify up to three values for the DNS servers used to resolve IP addresses into host names and vice versa. Ideally, configure a name server on your machine or somewhere else in your network before setting up DHCP. That name server should also define a host name for each dynamic address and vice versa. To learn how to configure your own name server, read *Chapter 23, The Domain Name System*.
- The line option broadcast-address defines the broadcast address the requesting client should use.
- With option routers, set where the server should send data packets that cannot be delivered to a host on the local network (according to the source and target host address and the subnet mask provided). Usually, especially in smaller networks, this router is identical to the Internet gateway.
- With option subnet-mask, specify the netmask assigned to clients.

The last section of the file defines a network, including a subnet mask. To finish, specify the address range that the DHCP daemon should use to assign IP addresses to interested clients. In *Example 24.1, "The Configuration File `/etc/dhcpd.conf`"*, clients may be given any address between 192.168.2.10 and 192.168.2.20 or 192.168.2.100 and 192.168.2.200.

After editing these few lines, you should be able to activate the DHCP daemon with the command **`systemctl start dhcpd`**. It will be ready for use immediately. Use the command **`rcd-hcpd check-syntax`** to perform a brief syntax check. If you encounter any unexpected problems with your configuration (the server aborts with an error or does not return done on start), you should be able to find out what has gone wrong by looking for information either in the main system log that can be queried with the command **`journalctl`** (see *Chapter 14, `journalctl`: Query the `systemd` Journal* for more information).

On a default SUSE Linux Enterprise Server system, the DHCP daemon is started in a chroot environment for security reasons. The configuration files must be copied to the chroot environment so the daemon can find them. Normally, there is no need to worry about this because the command `systemctl start dhcpd` automatically copies the files.

24.3.1 Clients with Fixed IP Addresses

DHCP can also be used to assign a predefined, static address to a specific client. Addresses assigned explicitly always take priority over dynamic addresses from the pool. A static address never expires in the way a dynamic address would, for example, if there were not enough addresses available and the server needed to redistribute them among clients.

To identify a client configured with a static address, `dhcpd` uses the hardware address (which is a globally unique, fixed numerical code consisting of six octet pairs) for the identification of all network devices (for example, `00:30:6E:08:EC:80`). If the respective lines, like the ones in *Example 24.2, “Additions to the Configuration File”*, are added to the configuration file of *Example 24.1, “The Configuration File `/etc/dhcpd.conf`”*, the DHCP daemon always assigns the same set of data to the corresponding client.

EXAMPLE 24.2: ADDITIONS TO THE CONFIGURATION FILE

```
host jupiter {  
  hardware ethernet 00:30:6E:08:EC:80;  
  fixed-address 192.168.2.100;  
}
```

The name of the respective client (`host host name`, here `jupiter`) is entered in the first line and the MAC address in the second line. On Linux hosts, find the MAC address with the command `ip link show` followed by the network device (for example, `eth0`). The output should contain something like

```
link/ether 00:30:6E:08:EC:80
```

In the preceding example, a client with a network card having the MAC address `00:30:6E:08:EC:80` is assigned the IP address `192.168.2.100` and the host name `jupiter` automatically. The type of hardware to enter is `ethernet` in nearly all cases, although `token-ring`, which is often found on IBM systems, is also supported.

24.3.2 The SUSE Linux Enterprise Server Version

To improve security, the SUSE Linux Enterprise Server version of the ISC's DHCP server comes with the non-root/chroot patch by Ari Edelkind applied. This enables `dhcpcd` to run with the user ID `nobody` and run in a chroot environment (`/var/lib/dhcp`). To make this possible, the configuration file `dhcpcd.conf` must be located in `/var/lib/dhcp/etc`. The init script automatically copies the file to this directory when starting.

Control the server's behavior regarding this feature by means of entries in the file `/etc/sysconfig/dhcpcd`. To run `dhcpcd` without the chroot environment, set the variable `DHCPD_RUN_CHROOTED` in `/etc/sysconfig/dhcpcd` to "no".

To enable `dhcpcd` to resolve host names even from within the chroot environment, some other configuration files must be copied as well:

- `/etc/localtime`
- `/etc/host.conf`
- `/etc/hosts`
- `/etc/resolv.conf`

These files are copied to `/var/lib/dhcp/etc/` when starting the init script. Take these copies into account for any changes that they require if they are dynamically modified by scripts like `/etc/ppp/ip-up`. However, there should be no need to worry about this if the configuration file only specifies IP addresses (instead of host names).

If your configuration includes additional files that should be copied into the chroot environment, set these under the variable `DHCPD_CONF_INCLUDE_FILES` in the file `/etc/sysconfig/dhcpcd`. To ensure that the DHCP logging facility keeps working even after a restart of the syslog daemon, there is an additional entry `SYSLOGD_ADDITIONAL_SOCKET_DHCP` in the file `/etc/sysconfig/syslog`.

24.4 For More Information

More information about DHCP is available at the Web site of the *Internet Systems Consortium* (<http://www.isc.org/products/DHCP/>). Information is also available in the `dhcpcd`, `dhcpcd.conf`, `dhcpcd.leases`, and `dhcp-options` man pages.

25 Sharing File Systems with NFS

Distributing and sharing file systems over a network is a common task in corporate environments. The well-proven network file system (*NFS*) works with *NIS*, the yellow pages protocol. For a more secure protocol that works with *LDAP* and Kerberos, check *NFSv4* (default). Combined with *pNFS*, you can eliminate performance bottlenecks.

NFS with NIS makes a network transparent to the user. With NFS, it is possible to distribute arbitrary file systems over the network. With an appropriate setup, users always find themselves in the same environment regardless of the terminal they currently use.



Important: Need for DNS

In principle, all exports can be made using IP addresses only. To avoid time-outs, you need a working DNS system. DNS is necessary at least for logging purposes, because the *mountd* daemon does reverse lookups.

25.1 Terminology

The following are terms used in the YaST module.

Exports

A directory *exported* by an NFS server, which clients can integrate it into their system.

NFS Client

The NFS client is a system that uses NFS services from an NFS server over the Network File System protocol. The TCP/IP protocol is already integrated into the Linux kernel; there is no need to install any additional software.

NFS Server

The NFS server provides NFS services to clients. A running server depends on the following daemons: *nfsd* (worker), *idmapd* (user and group name mappings to IDs and vice versa), *statd* (file locking), and *mountd* (mount requests).

NFSv3

NFSv3 is the version 3 implementation, the “old” stateless NFS that supports client authentication.

NFSv4

NFSv4 is the new version 4 implementation that supports secure user authentication via kerberos. NFSv4 requires one single port only and thus is better suited for environments behind a firewall than NFSv3.

The protocol is specified as <http://tools.ietf.org/html/rfc3530> .

pNFS

Parallel NFS, a protocol extension of NFSv4. Any pNFS clients can directly access the data on an NFS server.

25.2 Installing NFS Server

The NFS server software is not part of the default installation. If you configure an NFS server as described in *Section 25.3, “Configuring NFS Server”* you will automatically be prompted to install the required packages. Alternatively, install the package `nfs-kernel-server` with YaST or Zypper. Like NIS, NFS is a client/server system. However, a machine can be both—it can supply file systems over the network (export) and mount file systems from other hosts (import).



Note: Mounting NFS Volumes Locally on the Exporting Server

Mounting NFS volumes locally on the exporting server is not supported on SUSE Linux Enterprise systems, as is the case on all Enterprise-class Linux systems.

25.3 Configuring NFS Server

Configuring an NFS server can be done either through YaST or manually. For authentication, NFS can also be combined with Kerberos.

25.3.1 Exporting File Systems with YaST

With YaST, turn a host in your network into an NFS server—a server that exports directories and files to all hosts granted access to it or to all members of a group. Thus, the server can also provide applications without installing the applications locally on every host.

To set up such a server, proceed as follows:

PROCEDURE 25.1: SETTING UP AN NFS SERVER

1. Start YaST and select *Network Services > NFS Server*; see *Figure 25.1, “NFS Server Configuration Tool”*. You may be prompted to install additional software.

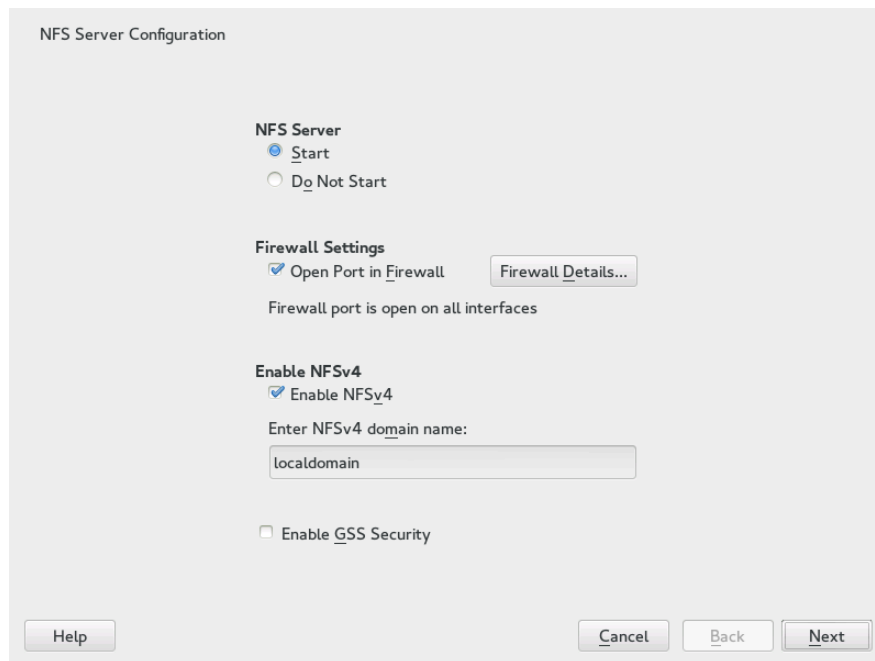


FIGURE 25.1: NFS SERVER CONFIGURATION TOOL

2. Activate the *Start* radio button.
3. If a firewall is active on your system (SuSEFirewall2), check *Open Ports in Firewall*. YaST adapts its configuration for the NFS server by enabling the `nfs` service.
4. Check whether you want to *Enable NFSv4*. If you deactivate NFSv4, YaST will only support NFSv3. For information about enabling NFSv2, see *Note: NFSv2*.
 - If NFSv4 is selected, additionally enter the appropriate NFSv4 domain name. Make sure the name is the same as the one in the `/etc/ldapd.conf` file of any NFSv4 client that accesses this particular server. This parameter is for the `ldapd` daemon that is required for NFSv4 support (on both server and client). Leave it as `localdomain` (the default) if you do not have any special requirements.

5. Click *Enable GSS Security* if you need secure access to the server. A prerequisite for this is to have Kerberos installed on your domain and to have both the server and the clients kerberized. Click *Next* to proceed with the next configuration dialog.
6. Click *Add Directory* in the upper half of the dialog to export your directory.
7. If you have not configured the allowed hosts already, another dialog for entering the client information and options pops up automatically. Enter the host wild card (usually you can leave the default settings as they are).

There are four possible types of host wild cards that can be set for each host: a single host (name or IP address), netgroups, wild cards (such as `*` indicating all machines can access the server), and IP networks.

For more information about these options, see the `exports` man page.
8. Click *Finish* to complete the configuration.

25.3.2 Exporting File Systems Manually

The configuration files for the NFS export service are `/etc/exports` and `/etc/sysconfig/nfs`. In addition to these files, `/etc/idmapd.conf` is needed for the NFSv4 server configuration. To start or restart the services, run the command `systemctl restart nfsserver`. This also starts the `rpc.idmapd` if NFSv4 is configured in `/etc/sysconfig/nfs`. The NFS server depends on a running RPC portmapper. Therefore, it also starts or restarts the portmapper service.



Note: NFSv4

NFSv4 is the latest version of NFS protocol available on SUSE Linux Enterprise Server. Configuring directories for export with NFSv4 is now the same as with NFSv3.

On the previous SUSE Linux Enterprise Server 11 version, the bind mount in `/etc/exports` was mandatory. It is still supported, but now deprecated.

`/etc/exports`

The `/etc/exports` file contains a list of entries. Each entry indicates a directory that is shared and how it is shared. A typical entry in `/etc/exports` consists of:

```
/shared/directory  host(option_list)
```

For example:

```
/export/data 192.168.1.2(rw, sync)
```

Here the IP address 192.168.1.2 is used to identify the allowed client. You can also use the name of the host, a wild card indicating a set of hosts (*.abc.com, *, etc.), or netgroups (@my-hosts).

For a detailed explanation of all options and their meaning, refer to the man page of **exports** (man exports).

/etc/sysconfig/nfs

The /etc/sysconfig/nfs file contains a few parameters that determine NFSv4 server daemon behavior. It is important to set the parameter NFS4_SUPPORT to yes (default). NFS4_SUPPORT determines whether the NFS server supports NFSv4 exports and clients.



Tip: Mount Options

On SUSE Linux Enterprise prior to version 12, the --bind mount in /etc/exports was mandatory. It is still supported, but now deprecated. Configuring directories for export with NFSv4 is now the same as with NFSv3.



Note: NFSv2

If NFS clients still depend on NFSv2, enable it on the server in /etc/sysconfig/nfs by setting:

```
NFSD_OPTIONS="-V2"  
MOUNTD_OPTIONS="-V2"
```

After restarting the service, check whether version 2 is available with the command:

```
tux > cat /proc/fs/nfsd/versions  
+2 +3 +4 +4.1 -4.2
```

/etc/idmapd.conf

Every user on a Linux machine has a name and an ID. `idmapd` does the name-to-ID mapping for NFSv4 requests to the server and replies to the client. It must be running on both server and client for NFSv4, because NFSv4 uses only names for its communication.

Make sure that there is a uniform way in which user names and IDs (uid) are assigned to users across machines that might probably be sharing file systems using NFS. This can be achieved by using NIS, LDAP, or any uniform domain authentication mechanism in your domain.

The parameter Domain must be set the same for both, client and server in the /etc/idmapd.conf file. If you are not sure, leave the domain as localdomain in the server and client files. A sample configuration file looks like the following:

```
[General]
Verbosity = 0
Pipefs-Directory = /var/lib/nfs/rpc_pipefs
Domain = localdomain

[Mapping]
Nobody-User = nobody
Nobody-Group = nobody
```

For more information, see the man pages of idmapd and idmapd.conf (man idmapd and man idmapd.conf).

After changing /etc/exports or /etc/sysconfig/nfs, start or restart the NFS server service:

```
systemctl restart nfsserver
```

After changing /etc/idmapd.conf, reload the configuration file:

```
killall -HUP rpc.idmapd
```

If the NFS service needs to start at boot time, run:

```
systemctl enable nfsserver
```

25.3.3 NFS with Kerberos

To use Kerberos authentication for NFS, GSS security must be enabled. Select *Enable GSS Security* in the initial YaST NFS Server dialog. You must have a working Kerberos server to use this feature. YaST does not set up the server but only uses the provided functionality. If you want to use Kerberos authentication in addition to the YaST configuration, complete at least the following steps before running the NFS configuration:

1. Make sure that both the server and the client are in the same Kerberos domain. They must access the same KDC (Key Distribution Center) server and share their `krb5.keytab` file (the default location on any machine is `/etc/krb5.keytab`). For more information about Kerberos, see *Book “Security Guide”, Chapter 7 “Network Authentication with Kerberos”*.
2. Start the gssd service on the client with `systemctl start rpc-gssd.service`.
3. Start the svcgssd service on the server with `systemctl start rpc-svcgssd.service`.

For more information about configuring kerberized NFS, refer to the links in [Section 25.5, “For More Information”](#).

25.4 Configuring Clients

To configure your host as an NFS client, you do not need to install additional software. All needed packages are installed by default.

25.4.1 Importing File Systems with YaST

Authorized users can mount NFS directories from an NFS server into the local file tree using the YaST NFS client module. Proceed as follows:

PROCEDURE 25.2: IMPORTING NFS DIRECTORIES

1. Start the YaST NFS client module.
2. Click *Add* in the *NFS Shares* tab. Enter the host name of the NFS server, the directory to import, and the mount point at which to mount this directory locally.
3. When using NFSv4, select *Enable NFSv4* in the *NFS Settings* tab. Additionally, the *NFSv4 Domain Name* must contain the same value as used by the NFSv4 server. The default domain is `localdomain`.
4. To use Kerberos authentication for NFS, GSS security must be enabled. Select *Enable GSS Security*.
5. Enable *Open Port in Firewall* in the *NFS Settings* tab if you use a Firewall and want to allow access to the service from remote computers. The firewall status is displayed next to the check box.

6. Click *OK* to save your changes.

The configuration is written to `/etc/fstab` and the specified file systems are mounted. When you start the YaST configuration client at a later time, it also reads the existing configuration from this file.



Tip: NFS as a Root File System

On (diskless) systems where the root partition is mounted via network as an NFS share, you need to be careful when configuring the network device with which the NFS share is accessible.

When shutting down or rebooting the system, the default processing order is to turn off network connections, then unmount the root partition. With NFS root, this order causes problems as the root partition cannot be cleanly unmounted as the network connection to the NFS share is already not activated. To prevent the system from deactivating the relevant network device, open the network device configuration tab as described in [Section 15.4.1.2.5, “Activating the Network Device”](#), and choose *On NFSroot* in the *Device Activation* pane.

25.4.2 Importing File Systems Manually

The prerequisite for importing file systems manually from an NFS server is a running RPC port mapper. The `nfs` service takes care to start it properly; thus, start it by entering **`systemctl start nfs`** as `root`. Then remote file systems can be mounted in the file system like local partitions using **`mount`**:

```
mount host:remote-pathlocal-path
```

To import user directories from the `nfs.example.com` machine, for example, use:

```
mount nfs.example.com:/home /home
```

25.4.2.1 Using the Automount Service

The `autofs` daemon can be used to mount remote file systems automatically. Add the following entry to the `/etc/auto.master` file:

```
/nfsmounts /etc/auto.nfs
```

Now the `/nfsmounts` directory acts as the root for all the NFS mounts on the client if the `auto.nfs` file is filled appropriately. The name `auto.nfs` is chosen for the sake of convenience—you can choose any name. In `auto.nfs` add entries for all the NFS mounts as follows:

```
localdata -fstype=nfs server1:/data
nfs4mount -fstype=nfs4 server2:/
```

Activate the settings with `systemctl start autofs` as `root`. In this example, `/nfsmounts/localdata`, the `/data` directory of `server1`, is mounted with NFS and `/nfsmounts/nfs4mount` from `server2` is mounted with NFSv4.

If the `/etc/auto.master` file is edited while the service `autofs` is running, the automounter must be restarted for the changes to take effect with `systemctl restart autofs`.

25.4.2.2 Manually Editing `/etc/fstab`

A typical NFSv3 mount entry in `/etc/fstab` looks like this:

```
nfs.example.com:/data /local/path nfs rw,noauto 0 0
```

For NFSv4 mounts, use `nfs4` instead of `nfs` in the third column:

```
nfs.example.com:/data /local/pathv4 nfs4 rw,noauto 0 0
```

The `noauto` option prevents the file system from being mounted automatically at start-up. If you want to mount the respective file system manually, it is possible to shorten the mount command specifying the mount point only:

```
mount /local/path
```



Note: Mounting at Start-Up

If you do not enter the `noauto` option, the init scripts of the system will handle the mount of those file systems at start-up.

25.4.3 Parallel NFS (pNFS)

NFS is one of the oldest protocols, developed in the '80s. As such, NFS is usually sufficient if you want to share small files. However, when you want to transfer big files or large numbers of clients want to access data, an NFS server becomes a bottleneck and significantly impacts on the system performance. This is because of files quickly getting bigger, whereas the relative speed of your Ethernet has not fully kept up.

When you request a file from a “normal” NFS server, the server looks up the file metadata, collects all the data and transfers it over the network to your client. However, the performance bottleneck becomes apparent no matter how small or big the files are:

- With small files most of the time is spent collecting the metadata.
- With big files most of the time is spent on transferring the data from server to client.

pNFS, or parallel NFS, overcomes this limitation as it separates the file system metadata from the location of the data. As such, pNFS requires two types of servers:

- A *metadata or control server* that handles all the non-data traffic
- One or more *storage server(s)* that hold(s) the data

The metadata and the storage servers form a single, logical NFS server. When a client wants to read or write, the metadata server tells the NFSv4 client which storage server to use to access the file chunks. The client can access the data directly on the server.

SUSE Linux Enterprise supports pNFS on the client side only.

25.4.3.1 Configuring pNFS Client With YaST

Proceed as described in [Procedure 25.2, “Importing NFS Directories”](#), but click the *pNFS (v4.1)* checkbox and optionally *NFSv4 share*. YaST will do all the necessary steps and will write all the required options in the file `/etc/exports`.

25.4.3.2 Configuring pNFS Client Manually

Refer to *Section 25.4.2, “Importing File Systems Manually”* to start. Most of the configuration is done by the NFSv4 server. For pNFS, the only difference is to add the `minorversion` option and the metadata server `MDS_SERVER` to your `mount` command:




```
mount -t nfs4 -o minorversion=1 MDS_SERVER MOUNTPOINT
```

To help with debugging, change the value in the `/proc` file system:

```
echo 32767 > /proc/sys/sunrpc/nfsd_debug
echo 32767 > /proc/sys/sunrpc/nfs_debug
```

25.5 For More Information

In addition to the man pages of `exports`, `nfs`, and `mount`, information about configuring an NFS server and client is available in `/usr/share/doc/packages/nfsidmap/README`. For further documentation online refer to the following Web sites:

- Find the detailed technical documentation online at [SourceForge \(http://nfs.sourceforge.net/\)](http://nfs.sourceforge.net/) .
- For instructions for setting up kerberized NFS, refer to [NFS Version 4 Open Source Reference Implementation \(http://www.citi.umich.edu/projects/nfsv4/linux/krb5-setup.html\)](http://www.citi.umich.edu/projects/nfsv4/linux/krb5-setup.html) .
- If you have questions on NFSv4, refer to the [Linux NFSv4 FAQ \(http://www.citi.umich.edu/projects/nfsv4/linux/faq/\)](http://www.citi.umich.edu/projects/nfsv4/linux/faq/) .

26 Samba

Using Samba, a Unix machine can be configured as a file and print server for macOS, Windows, and OS/2 machines. Samba has developed into a fully-fledged and rather complex product. Configure Samba with YaST, or by editing the configuration file manually.

26.1 Terminology

The following are some terms used in Samba documentation and in the YaST module.

SMB protocol

Samba uses the SMB (server message block) protocol that is based on the NetBIOS services. Microsoft released the protocol so other software manufacturers could establish connections to a Microsoft domain network. With Samba, the SMB protocol works on top of the TCP/IP protocol, so the TCP/IP protocol must be installed on all clients.



Tip: IBM z Systems: NetBIOS Support

IBM z Systems merely supports SMB over TCP/IP. NetBIOS support is not available on these systems.

CIFS protocol

CIFS (common Internet file system) protocol is another protocol supported by Samba. CIFS defines a standard remote file system access protocol for use over the network, enabling groups of users to work together and share documents across the network.

NetBIOS

NetBIOS is a software interface (API) designed for communication between machines providing a name service. It enables machines connected to the network to reserve names for themselves. After reservation, these machines can be addressed by name. There is no central process that checks names. Any machine on the network can reserve as many names as it wants as long as the names are not already in use. The NetBIOS interface can be implemented for different network architectures. An implementation that works relatively closely with network hardware is called NetBEUI, but this is often called NetBIOS. Network protocols implemented with NetBIOS are IPX from Novell (NetBIOS via TCP/IP) and TCP/IP.

The NetBIOS names sent via TCP/IP have nothing in common with the names used in /etc/hosts or those defined by DNS. NetBIOS uses its own, completely independent naming convention. However, it is recommended to use names that correspond to DNS host names to make administration easier or use DNS natively. This is the default used by Samba.

Samba server

Samba server provides SMB/CIFS services and NetBIOS over IP naming services to clients. For Linux, there are three daemons for Samba server: `smbd` for SMB/CIFS services, `nmbd` for naming services, and `winbind` for authentication.

Samba client

The Samba client is a system that uses Samba services from a Samba server over the SMB protocol. All common operating systems, such as macOS, Windows, and OS/2, support the SMB protocol. The TCP/IP protocol must be installed on all computers. Samba provides a client for the different Unix flavors. For Linux, there is a kernel module for SMB that allows the integration of SMB resources on the Linux system level. You do not need to run any daemon for the Samba client.

Shares

SMB servers provide resources to the clients by means of shares. Shares are printers and directories with their subdirectories on the server. It is exported by means of a name and can be accessed by its name. The share name can be set to any name—it does not need to be the name of the export directory. A printer is also assigned a name. Clients can access the printer by its name.

DC

A domain controller (DC) is a server that handles accounts in a domain. For data replication, additional domain controllers are available in one domain.

26.2 Installing a Samba Server

To install a Samba server, start YaST and select *Software > Software Management*. Choose *View > Patterns* and select *File Server*. Confirm the installation of the required packages to finish the installation process.

26.3 Starting and Stopping Samba

You can start or stop the Samba server automatically (during boot) or manually. Starting and stopping policy is a part of the YaST Samba server configuration described in [Section 26.4.1, “Configuring a Samba Server with YaST”](#).

From a command line, stop services required for Samba with `systemctl stop smb nmb` and start them with `systemctl start nmb smb`. The `smb` service cares about `winbind` if needed.



Tip: winbind

`winbind` is an independent service, and as such is also offered as an individual `samba-winbind` package.

26.4 Configuring a Samba Server

A Samba server in SUSE® Linux Enterprise Server can be configured in two different ways: with YaST or manually. Manual configuration offers a higher level of detail, but lacks the convenience of the YaST GUI.

26.4.1 Configuring a Samba Server with YaST

To configure a Samba server, start YaST and select *Network Services > Samba Server*.

26.4.1.1 Initial Samba Configuration

When starting the module for the first time, the *Samba Installation* dialog starts, prompting you to make a few basic decisions concerning administration of the server. At the end of the configuration it prompts for the Samba administrator password (*Samba Root Password*). For later starts, the *Samba Configuration* dialog appears.

The *Samba Installation* dialog consists of two steps and optional detailed settings:

Workgroup or Domain Name

Select an existing name from *Workgroup or Domain Name* or enter a new one and click *Next*.

Samba Server Type

In the next step, specify whether your server should act as a primary domain controller (PDC), backup domain controller (BDC), or not act as a domain controller. Continue with *Next*.

If you do not want to proceed with a detailed server configuration, confirm with *OK*. Then in the final pop-up box, set the *Samba root Password*.

You can change all settings later in the *Samba Configuration* dialog with the *Start-Up*, *Shares*, *Identity*, *Trusted Domains*, and *LDAP Settings* tabs.

26.4.1.2 Advanced Samba Configuration

During the first start of the Samba server module the *Samba Configuration* dialog appears directly after the two initial steps described in [Section 26.4.1.1, “Initial Samba Configuration”](#). Use it to adjust your Samba server configuration.

After editing your configuration, click *OK* to save your settings.

26.4.1.2.1 Starting the Server

In the *Start Up* tab, configure the start of the Samba server. To start the service every time your system boots, select *During Boot*. To activate manual start, choose *Manually*. More information about starting a Samba server is provided in [Section 26.3, “Starting and Stopping Samba”](#).

In this tab, you can also open ports in your firewall. To do so, select *Open Port in Firewall*. If you have multiple network interfaces, select the network interface for Samba services by clicking *Firewall Details*, selecting the interfaces, and clicking *OK*.

26.4.1.2.2 Shares

In the *Shares* tab, determine the Samba shares to activate. There are some predefined shares, like homes and printers. Use *Toggle Status* to switch between *Active* and *Inactive*. Click *Add* to add new shares and *Delete* to delete the selected share.

Allow Users to Share Their Directories enables members of the group in *Permitted Group* to share directories they own with other users. For example, users for a local scope or DOMAIN\Users for a domain scope. The user also must make sure that the file system permissions allow access. With *Maximum Number of Shares*, limit the total amount of shares that may be created. To permit access to user shares without authentication, enable *Allow Guest Access*.

26.4.1.2.3 Identity

In the *Identity* tab, you can determine the domain with which the host is associated (*Base Settings*) and whether to use an alternative host name in the network (*NetBIOS Hostname*). It is also possible to use Microsoft Windows Internet Name Service (WINS) for name resolution. In this case, activate *Use WINS for Hostname Resolution* and decide whether to *Retrieve WINS server via DHCP*. To set expert global settings or set a user authentication source, for example LDAP instead of TDB database, click *Advanced Settings*.

26.4.1.2.4 Trusted Domains

To enable users from other domains to access your domain, make the appropriate settings in the *Trusted Domains* tab. To add a new domain, click *Add*. To remove the selected domain, click *Delete*.

26.4.1.2.5 LDAP Settings

In the tab *LDAP Settings*, you can determine the LDAP server to use for authentication. To test the connection to your LDAP server, click *Test Connection*. To set expert LDAP settings or use default values, click *Advanced Settings*.

For more information about LDAP configuration, see Book “Security Guide”, Chapter 5 “LDAP —A Directory Service”.

26.4.2 Configuring the Server Manually

If you intend to use Samba as a server, install `samba`. The main configuration file for Samba is `/etc/samba/smb.conf`. This file can be divided into two logical parts. The `[global]` section contains the central and global settings. The following default sections contain the individual file and printer shares:

- `[homes]`
- `[profiles]`
- `[users]`
- `[groups]`
- `[printers]`
- `[print$]`

By means of this approach, details regarding the shares can be set differently or globally in the `[global]` section, which enhances the structural transparency of the configuration file.

26.4.2.1 The global Section

The following parameters of the `[global]` section need some adjustment to match the requirements of your network setup so other machines can access your Samba server via SMB in a Windows environment.

`workgroup = WORKGROUP`

This line assigns the Samba server to a workgroup. Replace `WORKGROUP` with an appropriate workgroup of your networking environment. Your Samba server appears under its DNS name unless this name has been assigned to some other machine in the network. If the DNS name is not available, set the server name using `netbiosname=MYNAME`. For more details about this parameter, see the `smb.conf` man page.

`os level = 20`

This parameter triggers whether your Samba server tries to become LMB (local master browser) for its workgroup. Choose a very low value such as `2` to spare the existing Windows network from any disturbances caused by a misconfigured Samba server. More in-

formation about this important topic can be found in the Network Browsing chapter of the Samba 3 Howto; for more information on the Samba 3 Howto, see [Section 26.9, “For More Information”](#).

If no other SMB server is in your network (such as a Windows 2000 server) and you want the Samba server to keep a list of all systems present in the local environment, set the `os_level` to a higher value (for example, `65`). Your Samba server is then chosen as LMB for your local network.

When changing this setting, consider carefully how this could affect an existing Windows network environment. First test the changes in an isolated network or at a noncritical time of day.

wins support and wins server

To integrate your Samba server into an existing Windows network with an active WINS server, enable the `wins_server` option and set its value to the IP address of that WINS server.

If your Windows machines are connected to separate subnets and need to still be aware of each other, you need to set up a WINS server. To turn a Samba server into such a WINS server, set the option `wins_support = Yes`. Make sure that only one Samba server of the network has this setting enabled. The options `wins_server` and `wins_support` must never be enabled at the same time in your `smb.conf` file.

26.4.2.2 Shares

The following examples illustrate how a CD-ROM drive and the user directories (`homes`) are made available to the SMB clients.

[cdrom]

To avoid having the CD-ROM drive accidentally made available, these lines are deactivated with comment marks (semicolons in this case). Remove the semicolons in the first column to share the CD-ROM drive with Samba.

EXAMPLE 26.1: A CD-ROM SHARE

```
[cdrom]
    comment = Linux CD-ROM
    path = /media/cdrom
    locking = No
```


[cdrom] and comment

The [cdrom] section entry is the name of the share that can be seen by all SMB clients on the network. An additional comment can be added to further describe the share.

path = /media/cdrom

path exports the directory /media/cdrom.

By means of a very restrictive default configuration, this kind of share is only made available to the users present on this system. If this share should be made available to everybody, add a line guest ok = yes to the configuration. This setting gives read permissions to anyone on the network. It is recommended to handle this parameter with great care. This applies even more to the use of this parameter in the [global] section.

[homes]

The [homes] share is of special importance here. If the user has a valid account and password for the Linux file server and his own home directory, he can be connected to it.

EXAMPLE 26.2: [HOMES] SHARE

```
[homes]
    comment = Home Directories
    valid users = %S
    browseable = No
    read only = No
    inherit acls = Yes
```

[homes]

As long as there is no other share using the share name of the user connecting to the SMB server, a share is dynamically generated using the [homes] share directives. The resulting name of the share is the user name.

valid users = %S

%S is replaced with the concrete name of the share when a connection has been successfully established. For a [homes] share, this is always the user name. As a consequence, access rights to a user's share are restricted exclusively to that user.

browseable = No

This setting makes the share invisible in the network environment.

read only = No

By default, Samba prohibits write access to any exported share by means of the read only = Yes parameter. To make a share writable, set the value read only = No, which is synonymous with writable = Yes.

create mask = 0640

Systems that are not based on MS Windows NT do not understand the concept of Unix permissions, so they cannot assign permissions when creating a file. The parameter create mask defines the access permissions assigned to newly created files. This only applies to writable shares. In effect, this setting means the owner has read and write permissions and the members of the owner's primary group have read permissions. valid users = %S prevents read access even if the group has read permissions. For the group to have read or write access, deactivate the line valid users = %S.

26.4.2.3 Security Levels

To improve security, each share access can be protected with a password. SMB offers the following ways of checking permissions:

User Level Security (security = user)

This variant introduces the concept of the user to SMB. Each user must register with the server with his or her own password. After registration, the server can grant access to individual exported shares dependent on user names.

ADS Level Security (security = ADS)

In this mode, Samba will act as a domain member in an Active Directory environment. To operate in this mode, the machine running Samba needs Kerberos installed and configured. You must join the machine using Samba to the ADS realm. This can be done using the *YaST Windows Domain Membership* module.

Domain Level Security (security = domain)

This mode will only work correctly if the machine has been joined into a Windows NT Domain. Samba will try to validate user name and password by passing it to a Windows NT Primary or Backup Domain Controller. The same way as a Windows NT Server would do. It expects the encrypted passwords parameter to be set to yes.

The selection of share, user, server, or domain level security applies to the entire server. It is not possible to offer individual shares of a server configuration with share level security and others with user level security. However, you can run a separate Samba server for each configured IP address on a system.

More information about this subject can be found in the Samba 3 HOWTO. For multiple servers on one system, pay attention to the options interfaces and bind interfaces only.

26.5 Configuring Clients

Clients can only access the Samba server via TCP/IP. NetBEUI and NetBIOS via IPX cannot be used with Samba.

26.5.1 Configuring a Samba Client with YaST

Configure a Samba client to access resources (files or printers) on the Samba or Windows server. Enter the NT or Active Directory domain or workgroup in the dialog *Network Services > Windows Domain Membership*. If you activate *Also Use SMB Information for Linux Authentication*, the user authentication runs over the Samba, NT or Kerberos server.

Click *Expert Settings* for advanced configuration options. For example, use the *Mount Server Directories* table to enable mounting server home directory automatically with authentication. This way users can access their home directories when hosted on CIFS. For details, see the pam_mount man page.

After completing all settings, confirm the dialog to finish the configuration.

26.6 Samba as Login Server

In networks where predominantly Windows clients are found, it is often preferable that users may only register with a valid account and password. In a Windows-based network, this task is handled by a primary domain controller (PDC). You can use a Windows NT server configured as PDC, but this task can also be done with a Samba server. The entries that must be made in the [global] section of smb.conf are shown in *Example 26.3, "Global Section in smb.conf"*.

EXAMPLE 26.3: GLOBAL SECTION IN SMB.CONF

```
[global]
    workgroup = WORKGROUP
    domain logons = Yes
    domain master = Yes
```

It is necessary to prepare user accounts and passwords in an encryption format that conforms with Windows. Do this with the command **smbpasswd -a name**. Create the domain account for the computers, required by the Windows domain concept, with the following commands:

```
useradd hostname\$$
smbpasswd -a -m hostname
```

With the **useradd** command, a dollar sign is added. The command **smbpasswd** inserts this automatically when the parameter **-m** is used. The commented configuration example (</usr/share/doc/packages/samba/examples/smb.conf.SUSE>) contains settings that automate this task.

```
add machine script = /usr/sbin/useradd -g nogroup -c "NT Machine Account" \
-s /bin/false %m\$$
```

To make sure that Samba can execute this script correctly, choose a Samba user with the required administrator permissions and add it to the **ntadmin** group. Then all users belonging to this Linux group can be assigned **Domain Admin** status with the command:

```
net groupmap add ntgroup="Domain Admins" unixgroup=ntadmin
```

26.7 Samba Server in the Network with Active Directory

If you run Linux servers and Windows servers together, you can build two independent authentication systems and networks or connect servers to one network with one central authentication system. Because Samba can cooperate with an active directory domain, you can join your SUSE Linux Enterprise Server to Active Directory (AD).

To join an AD domain proceed as follows:

1. Log in as `root` and start YaST.
2. Start *Network Services > Windows Domain Membership*.
3. Enter the domain to join at *Domain or Workgroup* in the *Windows Domain Membership* screen.

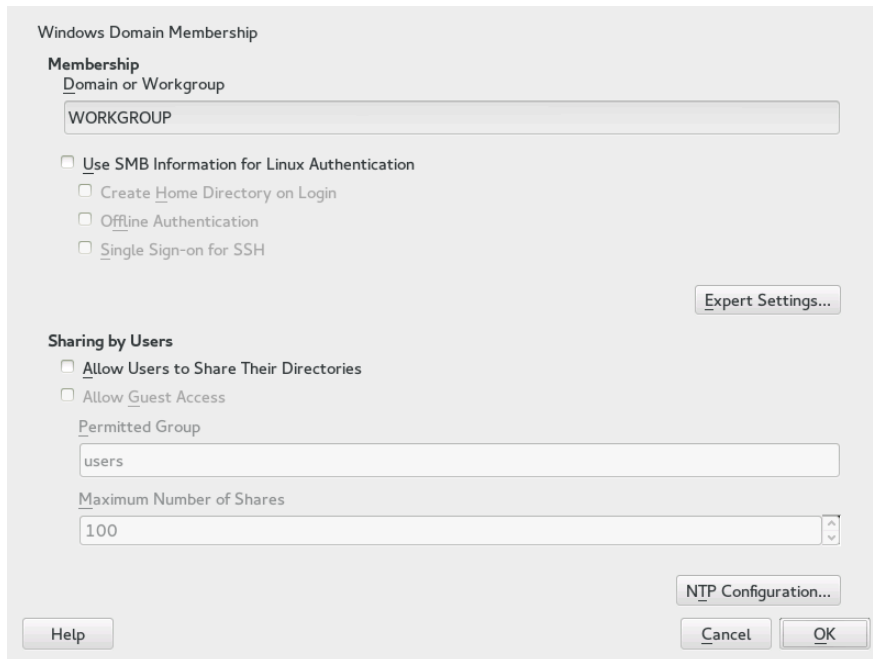


FIGURE 26.1: DETERMINING WINDOWS DOMAIN MEMBERSHIP

4. Check *Also Use SMB Information for Linux Authentication* to use the SMB source for Linux authentication on your SUSE Linux Enterprise Server.
5. Click *OK* and confirm the domain join when prompted for it.
6. Provide the password for the Windows Administrator on the AD server and click *OK*.
Your server is now set up to pull in all authentication data from the Active Directory domain controller.



Tip: Identity Mapping

In an environment with more than one Samba server, UIDs and GIDs will not be created consistently. The UIDs that get assigned to users will be dependent on the order in which they first log in, which results in UID conflicts across servers. To fix this, you need to use identity mapping. See <https://www.samba.org/samba/docs/man/Samba-HOWTO-Collection/idmapper.html> for more details.

26.8 Advanced Topics

This section introduces more advanced techniques to manage both the client and server part of the Samba suite.

26.8.1 Transparent File Compression on Btrfs

Samba allows clients to remotely manipulate file and directory compression flags for shares placed on the Btrfs file system. Windows Explorer provides the ability to flag files/directories for transparent compression via the *File > Properties > Advanced* dialog:

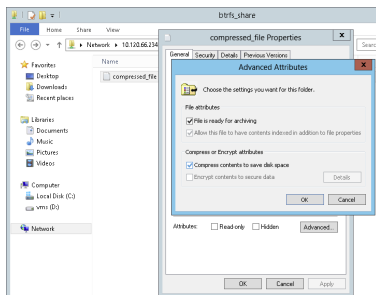


FIGURE 26.2: WINDOWS EXPLORER ADVANCED ATTRIBUTES DIALOG

Files flagged for compression are transparently compressed and decompressed by the underlying file system when accessed or modified. This normally results in storage capacity savings at the expense of extra CPU overhead when accessing the file. New files and directories inherit the compression flag from the parent directory, unless created with the `FILE_NO_COMPRESSION` option.

Windows Explorer presents compressed files and directories visually differently to those that are not compressed:

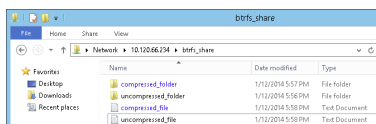


FIGURE 26.3: WINDOWS EXPLORER DIRECTORY LISTING WITH COMPRESSED FILES

You can enable Samba share compression either manually by adding

```
vfs objects = btrfs
```

to the share configuration in `/etc/samba/smb.conf`, or using YaST: *Network Services* > *Samba Server* > *Add*, and checking *Utilize Btrfs Features*.

A general overview about compression on Btrfs can be found in *Book “Storage Administration Guide”, Chapter 1 “Overview of File Systems in Linux”, Section 1.2.2.1 “Mounting Compressed Btrfs File Systems”*.

26.8.2 Snapshots

Snapshots, also called Shadow Copies, are copies of the state of a file system subvolume at a certain point of time. Snapper is the tool to manage these snapshots in Linux. Snapshots are supported on the Btrfs file system or thin-provisioned LVM volumes. The Samba suite supports managing of remote snapshots through the FSRVP protocol on both the server and client side.

26.8.2.1 Previous Versions

Snapshots on a Samba server can be exposed to remote Windows clients as file or directory previous versions.

To enable snapshots on a Samba server, the following conditions must be fulfilled:

- The SMB network share resides on a Btrfs subvolume.
- The SMB network share path has a related snapper configuration file. You can create the snapper file with

```
snapper -c <cfg_name> create-config /path/to/share
```

For more information on snapper, see *Chapter 6, System Recovery and Snapshot Management with Snapper*.

- The snapshot directory tree must allow access for relevant users. For more information, see the PERMISSIONS section of the `vfs_snapper` manual page (**man 8 vfs_snapper**).

To support remote snapshots, you need to modify the `/etc/samba/smb.conf` file. You can do it either with YaST > *Network Services* > *Samba Server*, or manually by enhancing the relevant share section with

```
vfs objects = snapper
```

Note that you need to restart the Samba service for manual `smb.conf` changes to take effect:

```
systemctl restart nmb smb
```

New Share

Identification

Share Name

Snapshotted Share

Share Description

Share Type

☐ Printer

☒ Directory

Share Path

/var/tmp

☐ Read-Only

☒ Inherit ACLs

☒ Expose Snapshots

☐ Utilize Btrfs Features

FIGURE 26.4: **ADDING A NEW SAMBA SHARE WITH SNAPSHOTTING ENABLED**

After being configured, snapshots created by snapper for the Samba share path can be accessed from Windows Explorer from a file or directory's *Previous Versions* tab.

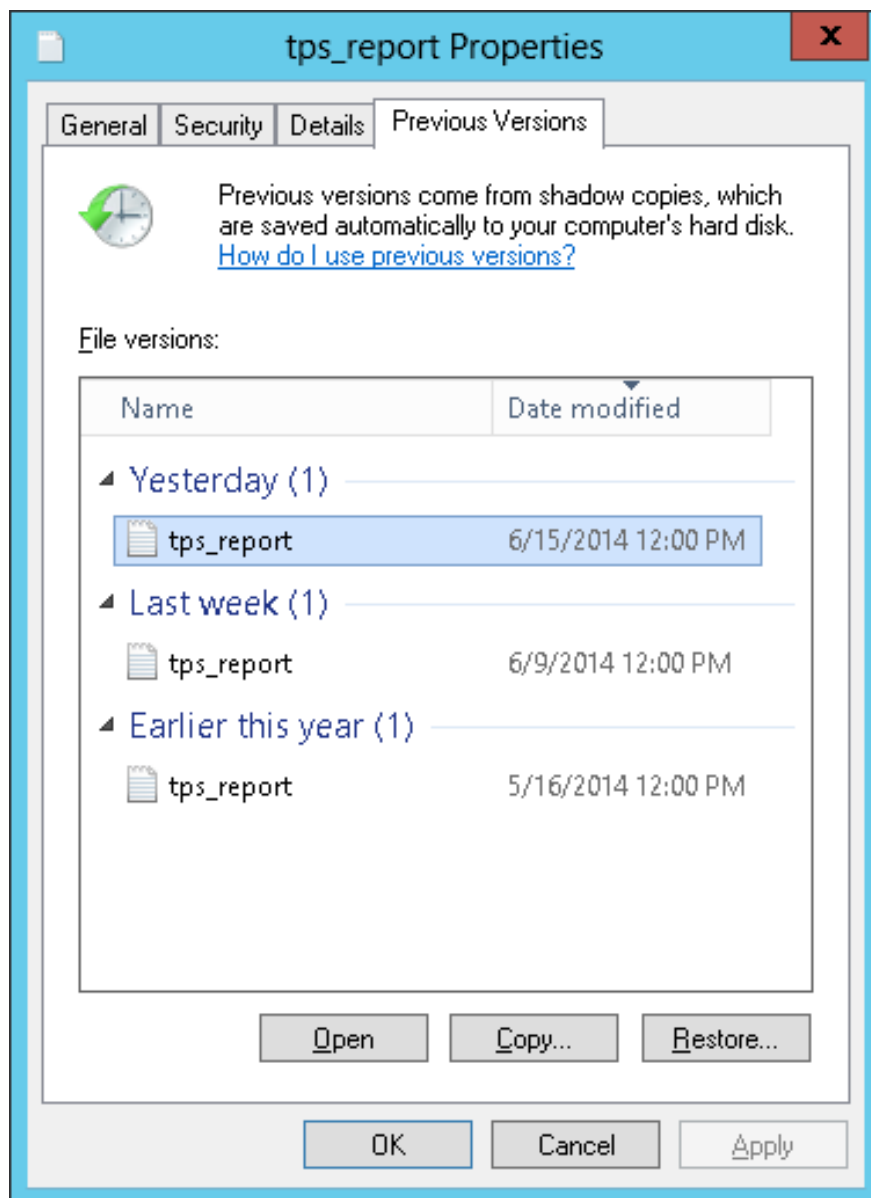


FIGURE 26.5: THE PREVIOUS VERSIONS TAB IN WINDOWS EXPLORER

26.8.2.2 Remote Share Snapshots

By default, snapshots can only be created and deleted on the Samba server locally, via the snapper command line utility, or using snapper's time line feature.

Samba can be configured to process share snapshot creation and deletion requests from remote hosts using the File Server Remote VSS Protocol (FSRVP).

In addition to the configuration and prerequisites documented in [Section 26.8.2.1, “Previous Versions”](#), the following global configuration is required in `/etc/samba/smb.conf`:

```
[global]
rpc_daemon:fssd = fork
registry shares = yes
include = registry
```

FSRVP clients, including Samba's `rpcclient` and Windows Server 2012 `DiskShadow.exe`, can then instruct Samba to create or delete a snapshot for a given share, and expose the snapshot as a new share.

26.8.2.3 Managing Snapshots Remotely from Linux with `rpcclient`

The `samba-client` package contains an FSRVP client that can remotely request a Windows/Samba server to create and expose a snapshot of a given share. You can then use existing tools in SUSE Linux Enterprise Server to mount the exposed share and back up its files. Requests to the server are sent using the `rpcclient` binary.

EXAMPLE 26.4: USING `rpcclient` TO REQUEST A WINDOWS SERVER 2012 SHARE SNAPSHOT

Connect to `win-server.example.com` server as an administrator in an `EXAMPLE` domain:

```
# rpcclient -U 'EXAMPLE\Administrator' ncacn_np:win-
server.example.com[ndr64,sign]
Enter EXAMPLE/Administrator's password:
```

Check that the SMB share is visible for `rpcclient`:

```
rpcclient $> netshareenum
netname: windows_server_2012_share
remark:
path:    C:\Shares\windows_server_2012_share
password:      (null)
```

Check that the SMB share supports snapshot creation:

```
rpcclient $> fss_is_path_sup windows_server_2012_share \
UNC \\WIN-SERVER\windows_server_2012_share\ supports shadow copy requests
```

Request the creation of a share snapshot:

```
rpcclient $> fss_create_expose backup ro windows_server_2012_share
```

```
13fe880e-e232-493d-87e9-402f21019fb6: shadow-copy set created
13fe880e-e232-493d-87e9-402f21019fb6(1c26544e-8251-445f-be89-d1e0a3938777): \
\\WIN-SERVER\windows_server_2012_share\ shadow-copy added to set
13fe880e-e232-493d-87e9-402f21019fb6: prepare completed in 0 secs
13fe880e-e232-493d-87e9-402f21019fb6: commit completed in 1 secs
13fe880e-e232-493d-87e9-402f21019fb6(1c26544e-8251-445f-be89-d1e0a3938777): \
share windows_server_2012_share@{1C26544E-8251-445F-BE89-D1E0A3938777} \
exposed as a snapshot of \\WIN-SERVER\windows_server_2012_share\
```

Confirm that the snapshot share is exposed by the server:

```
rpcclient $> netshareenum
netname: windows_server_2012_share
remark:
path: C:\Shares\windows_server_2012_share
password: (null)

netname: windows_server_2012_share@{1C26544E-8251-445F-BE89-D1E0A3938777}
remark: (null)
path: \\?\GLOBALROOT\Device\HarddiskVolumeShadowCopy{F6E6507E-F537-11E3-9404-
B8AC6F927453}\Shares\windows_server_2012_share\
password: (null)
```

Attempt to delete the snapshot share:

```
rpcclient $> fss_delete windows_server_2012_share \
13fe880e-e232-493d-87e9-402f21019fb6 1c26544e-8251-445f-be89-d1e0a3938777
13fe880e-e232-493d-87e9-402f21019fb6(1c26544e-8251-445f-be89-d1e0a3938777): \
\\WIN-SERVER\windows_server_2012_share\ shadow-copy deleted
```

Confirm that the snapshot share has been removed by the server:

```
rpcclient $> netshareenum
netname: windows_server_2012_share
remark:
path: C:\Shares\windows_server_2012_share
password: (null)
```

26.8.2.4 Managing Snapshots Remotely from Windows with **DiskShadow.exe**

You can manage snapshots of SMB shares on the Linux Samba server from the Windows environment acting as a client as well. Windows Server 2012 includes the **DiskShadow.exe** utility that can manage remote shares similar to the **rpcclient** described in [Section 26.8.2.3, “Managing Snapshots Remotely from Linux with **rpcclient**”](#). Note that you need to carefully set up the Samba server first.

Following is an example procedure to set up the Samba server so that the Windows Server client can manage its share's snapshots. Note that *EXAMPLE* is the Active Directory domain used in the testing environment, fsrvp-server.example.com is the host name of the Samba server, and /srv/smb is the path to the SMB share.

PROCEDURE 26.1: DETAILED SAMBA SERVER CONFIGURATION

1. Join Active Directory domain via YaST. For more information, [Section 26.7, “Samba Server in the Network with Active Directory”](#).

2. Ensure that the Active Domain DNS entry was correct:

```
fsrvp-server:~ # net -U 'Administrator' ads dns register \  
fsrvp-server.example.com <IP address>  
Successfully registered hostname with DNS
```

3. Create Btrfs subvolume at /srv/smb

```
fsrvp-server:~ # btrfs subvolume create /srv/smb
```

4. Create snapper configuration file for path /srv/smb

```
fsrvp-server:~ # snapper -c <snapper_config> create-config /srv/smb
```

5. Create new share with path /srv/smb, and YaST *Expose Snapshots* check box enabled. Make sure to add the following snippets to the global section of /etc/samba/smb.conf as mentioned in [Section 26.8.2.2, “Remote Share Snapshots”](#):

```
[global]  
rpc_daemon:fssd = fork  
registry shares = yes  
include = registry
```

6. Restart Samba with `systemctl restart nmb smb`

7. Configure snapper permissions:

```
fsrvp-server:~ # snapper -c <snapper_config> set-config \
ALLOW_USERS="EXAMPLE\\\\Administrator EXAMPLE\\\\win-client$"
```

Ensure that any ALLOW_USERS are also permitted traversal of the `.snapshots` subdirectory.

```
fsrvp-server:~ # snapper -c <snapper_config> set-config SYNC_ACL=yes
```

! Important: Path Escaping

Be careful about the `\` escapes! Escape twice to ensure that the value stored in `/etc/snapper/configs/<snapper_config>` is escaped once.

"EXAMPLE\win-client\$" corresponds to the Windows client computer account. Windows issues initial FSRVP requests while authenticated with this account.

8. Grant Windows client account necessary privileges:

```
fsrvp-server:~ # net -U 'Administrator' rpc rights grant \
"EXAMPLE\win-client$" SeBackupPrivilege
Successfully granted rights.
```

The previous command is not needed for the "EXAMPLE\Administrator" user, which has privileges already granted.

PROCEDURE 26.2: WINDOWS CLIENT SETUP AND DiskShadow.exe IN ACTION

1. Boot Windows Server 2012 (example host name WIN-CLIENT).
2. Join the same Active Directory domain EXAMPLE as with the SUSE Linux Enterprise Server.
3. Reboot.
4. Open Powershell.
5. Start **DiskShadow.exe** and begin the backup procedure:

```
PS C:\Users\Administrator.EXAMPLE> diskshadow.exe
```

```
Microsoft DiskShadow version 1.0
Copyright (C) 2012 Microsoft Corporation
On computer: WIN-CLIENT, 6/17/2014 3:53:54 PM

DISKSHADOW> begin backup
```

6. Specify that shadow copy persists across program exit, reset or reboot:

```
DISKSHADOW> set context PERSISTENT
```

7. Check whether the specified share supports snapshots, and create one:

```
DISKSHADOW> add volume \\fsrvp-server\sles_snapper

DISKSHADOW> create
Alias VSS_SHADOW_1 for shadow ID {de4ddca4-4978-4805-8776-cdf82d190a4a} set as \
environment variable.
Alias VSS_SHADOW_SET for shadow set ID {c58e1452-c554-400e-a266-d11d5c837cb1} \
set as environment variable.

Querying all shadow copies with the shadow copy set ID \
{c58e1452-c554-400e-a266-d11d5c837cb1}

* Shadow copy ID = {de4ddca4-4978-4805-8776-cdf82d190a4a}      %VSS_SHADOW_1%
  - Shadow copy set: {c58e1452-c554-400e-a266-d11d5c837cb1}  %VSS_SHADOW_SET%
  - Original count of shadow copies = 1
  - Original volume name: \\FSRVP-SERVER\SLES_SNAPPER\ \
    [volume not on this machine]
  - Creation time: 6/17/2014 3:54:43 PM
  - Shadow copy device name:
    \\FSRVP-SERVER\SLES_SNAPPER@{31afd84a-44a7-41be-b9b0-751898756faa}
  - Originating machine: FSRVP-SERVER
  - Service machine: win-client.example.com
  - Not exposed
  - Provider ID: {89300202-3cec-4981-9171-19f59559e0f2}
  - Attributes: No_Auto_Release Persistent FileShare

Number of shadow copies listed: 1
```

8. Finish the backup procedure:


```
DISKSHADOW> end backup
```

9. After the snapshot was created, try to delete it and verify the deletion:

```
DISKSHADOW> delete shadows volume \\FSRVP-SERVER\SLES_SNAPPER\  
Deleting shadow copy {de4ddca4-4978-4805-8776-cdf82d190a4a} on volume \  
\\FSRVP-SERVER\SLES_SNAPPER\ from provider \  
{89300202-3cec-4981-9171-19f59559e0f2} [Attributes: 0x04000009]...  
  
Number of shadow copies deleted: 1  
  
DISKSHADOW> list shadows all  
  
Querying all shadow copies on the computer ...  
No shadow copies found in system.
```

26.9 For More Information

Documentation for Samba ships with the `samba-doc` package which is not installed by default. Install it with `zypper install samba-doc`. Enter `apropos samba` at the command line to display some manual pages or browse the `/usr/share/doc/packages/samba` directory for more online documentation and examples. Find a commented example configuration (`smb.conf.SUSE`) in the `examples` subdirectory. Another file to look for Samba related information is `/usr/share/doc/packages/samba/README.SUSE`.

The Samba HOWTO (see <https://wiki.samba.org> ) provided by the Samba team includes a section about troubleshooting. In addition to that, Part V of the document provides a step-by-step guide to checking your configuration.

27 On-Demand Mounting with Autofs

autofs is a program that automatically mounts specified directories on an on-demand basis. It is based on a kernel module for high efficiency, and can manage both local directories and network shares. These automatic mount points are mounted only when they are accessed, and unmounted after a certain period of inactivity. This on-demand behavior saves bandwidth and results in better performance than static mounts managed by /etc/fstab. While autofs is a control script, automount is the command (daemon) that does the actual auto-mounting.

27.1 Installation

autofs is not installed on SUSE Linux Enterprise Server by default. To use its auto-mounting capabilities, first install it with

```
sudo zypper install autofs
```

27.2 Configuration

You need to configure autofs manually by editing its configuration files with a text editor, such as vim. There are two basic steps to configure autofs—the *master* map file, and specific map files.

27.2.1 The Master Map File

The default master configuration file for autofs is /etc/auto.master. You can change its location by changing the value of the DEFAULT_MASTER_MAP_NAME option in /etc/sysconfig/autofs. Here is the content of the default one for SUSE Linux Enterprise Server:

```
#
# Sample auto.master file
# This is an automounter map and it has the following format
# key [ -mount-options-separated-by-comma ] location
# For details of the format look at autofs(5). ①
```



```
#
#/misc /etc/auto.misc ❷
#/net -hosts
#
# Include /etc/auto.master.d/*.autofs ❸
#
#+dir:/etc/auto.master.d
#
# Include central master map if it can be found using
# nsswitch sources.
#
# Note that if there are entries for /net or /misc (as
# above) in the included master map any keys that are the
# same will not be seen as the first read key seen takes
# precedence.
#
+auto.master ❹
```

- ❶ The autofs manual page (man 5 autofs) offers a lot of valuable information on the format of the automounter maps.
- ❷ Although commented out (#) by default, this is an example of a simple automounter mapping syntax.
- ❸ In case you need to split the master map into several files, uncomment the line, and put the mappings (suffixed with .autofs) in the /etc/auto.master.d/ directory.
- ❹ +auto.master ensures that those using NIS (see *Book “Security Guide”, Chapter 3 “Using NIS”, Section 3.1 “Configuring NIS Servers”* for more information on NIS) will still find their master map.

Entries in auto.master have three fields with the following syntax:

mount point	map name	options
-------------	----------	---------

mount point

The base location where to mount the autofs file system, such as /home.

map name

The name of a map source to use for mounting. For the syntax of the maps files, see [Section 27.2.2, “Map Files”](#).

options

These options (if specified) will apply as defaults to all entries in the given map.



Tip: For More Information

For more detailed information on the specific values of the optional `map-type`, `format`, and `options`, see the *auto.master* manual page (**man 5 auto.master**).

The following entry in `auto.master` tells `autofs` to look in `/etc/auto.smb`, and create mount points in the `/smb` directory.

```
/smb    /etc/auto.smb
```

27.2.1.1 Direct Mounts

Direct mounts create a mount point at the path specified inside the relevant map file. Instead of specifying the mount point in `auto.master`, replace the mount point field with `/-`. For example, the following line tells `autofs` to create a mount point at the place specified in `auto.smb`:

```
/-      /etc/auto.smb
```



Tip: Maps without Full Path

If the map file is not specified with its full local or network path, it is located using the Name Service Switch (NSS) configuration:

```
/-      auto.smb
```

27.2.2 Map Files



Important: Other Types of Maps

Although *files* are the most common types of maps for auto-mounting with `autofs`, there are other types as well. A map specification can be the output of a command, or a result of a query in LDAP or database. For more detailed information on map types, see the manual page **man 5 auto.master**.

Map files specify the (local or network) source location, and the mount point where to mount the source locally. The general format of maps is similar to the master map. The difference is that the *options* appear between the mount point and the location instead of at the end of the entry:

mount point	options	location
-------------	---------	----------

mount point

Specifies where to mount the source location. This can be either a single directory name (so called *indirect* mount) to be added to the base mount point specified in `auto.master`, or the full path of the mount point (direct mount, see [Section 27.2.1.1, “Direct Mounts”](#)).

options

Specifies optional comma-separated list of mount options for the relevant entries. If `auto.master` contains options for this map file as well, these are appended.

location

Specifies from where the file system is to be mounted. It is usually an NFS or SMB volume in the usual notation `host_name:path_name`. If the file system to be mounted begins with a '/' (such as local `/dev` entries or smbfs shares), a colon symbol ':' needs to be prefixed, such as `:/dev/sda1`.

27.3 Operation and Debugging

This section introduces information on how to control the `autofs` service operation, and how to view more debugging information when tuning the automounter operation.

27.3.1 Controlling the autofs Service

The operation of the `autofs` service is controlled by `systemd`. The general syntax of the `systemctl` command for `autofs` is

```
sudo systemctl sub-command autofs
```

where `sub-command` is one of:

enable

Starts the automounter daemon at boot.

start

Starts the automounter daemon.

stop

Stops the automounter daemon. Automatic mount points are not accessible.

status

Prints the current status of the autofs service together with a part of a relevant log file.

restart

Stops and starts the automounter, terminating all running daemons and starting new ones.

reload

Checks the current auto.master map, restarts those daemons whose entries have changed, and starts new ones for new entries.

27.3.2 Debugging the Automounter Problems

If you experience problems when mounting directories with autofs, it is useful to run the **automount** daemon manually and watch its output messages:

1. Stop autofs.

```
sudo systemctl stop autofs
```

2. From one terminal, run **automount** manually in the foreground, producing verbose output.

```
sudo automount -f -v
```

3. From another terminal, try to mount the auto-mounting file systems by accessing the mount points (for example by cd or ls).
4. Check the output of **automount** from the first terminal for more information why the mount failed, or why it was not even attempted.

27.4 Auto-Mounting an NFS Share

The following procedure illustrates how to configure `autofs` to auto-mount an NFS share available on your network. It makes use of the information mentioned above, and assumes you are familiar with NFS exports. For more information on NFS, see [Chapter 25, Sharing File Systems with NFS](#).

1. Edit the master map file `/etc/auto.master`:

```
sudo vim /etc/auto.master
```

Add a new entry for the new NFS mount at the end of `/etc/auto.master`:

```
/nfs      /etc/auto.nfs      --timeout=10
```

It tells `autofs` that the base mount point is `/nfs`, the NFS shares are specified in the `/etc/auto.nfs` map, and that all shares in this map will be automatically unmounted after 10 seconds of inactivity.

2. Create a new map file for NFS shares:

```
sudo vim /etc/auto.nfs
```

`/etc/auto.nfs` normally contains a separate line for each NFS share. Its format is described in [Section 27.2.2, "Map Files"](#). Add the line describing the mount point and the NFS share network address:

```
export      jupiter.com:/home/geeko/doc/export
```

The above line means that the `/home/geeko/doc/export` directory on the `jupiter.com` host will be auto-mounted to the `/nfs/export` directory on the local host (`/nfs` is taken from the `auto.master` map) when requested. The `/nfs/export` directory will be created automatically by `autofs`.

3. Optionally comment out the related line in `/etc/fstab` if you previously mounted the same NFS share statically. The line should look similar to this:

```
#jupiter.com:/home/geeko/doc/export /nfs/export nfs defaults 0 0
```

4. Reload autofs and check if it works:

```
sudo systemctl restart autofs
```

```
# ls -l /nfs/export
total 20
drwxr-xr-x  6 1001 users 4096 Oct 25 08:56 ./
drwxr-xr-x  3 root  root   0 Apr  1 09:47 ../
drwxr-xr-x  5 1001 users 4096 Jan 14 2013 .images/
drwxr-xr-x 10 1001 users 4096 Aug 16 2013 .profiled/
drwxr-xr-x  3 1001 users 4096 Aug 30 2013 .tmp/
drwxr-xr-x  4 1001 users 4096 Oct 25 08:56 SLE-12-manual/
```

If you can see the list of files on the remote share, then autofs is functioning.

27.5 Advanced Topics

This section describes topics that are beyond the basic introduction to autofs—auto-mounting of NFS shares that are available on your network, using wild cards in map files, and information specific to the CIFS file system.

27.5.1 /net Mount Point

This helper mount point is useful if you use a lot of NFS shares. /net auto-mounts all NFS shares on your local network on demand. The entry is already present in the auto.master file, so all you need to do is uncomment it and restart autofs:

```
/net      -hosts
```

```
systemctl restart autofs
```

For example, if you have a server named jupiter with an NFS share called /export, you can mount it by typing

```
# cd /net/jupiter/export
```

on the command line.

27.5.2 Using Wild Cards to Auto-Mount Subdirectories

If you have a directory with subdirectories that you need to auto-mount individually—the typical case is the `/home` directory with individual users' home directories inside—then `autofs` has a handy solution for you.

In case of home directories, add the following line in `auto.master`:

```
/home      /etc/auto.home
```

Now you need to add the correct mapping to the `/etc/auto.home` file, so that the users' home directories are mounted automatically. One solution is to create separate entries for each directory:

```
wilber      jupiter.com:/home/wilber
penguin     jupiter.com:/home/penguin
tux         jupiter.com:/home/tux
[...]
```

This is very awkward as you need to manage the list of users inside `auto.home`. You can use the asterisk `*` instead of the mount point, and the ampersand `&` instead of the directory to be mounted:

```
*          jupiter:/home/&
```

27.5.3 Auto-Mounting CIFS File System

If you want to auto-mount an SMB/CIFS share (see [Chapter 26, Samba](#) for more information on the SMB/CIFS protocol), you need to modify the syntax of the map file. Add `-fstype=cifs` in the option field, and prefix the share location with a colon `!`.

```
mount point    -fstype=cifs      ://jupiter.com/export
```

28 SLP

Configuring a network client requires detailed knowledge about services provided over the network (such as printing or LDAP, for example). To make it easier to configure such services on a network client, the “service location protocol” (SLP) was developed. SLP makes the availability and configuration data of selected services known to all clients in the local network. Applications that support SLP can use this information to be configured automatically.

SUSE® Linux Enterprise Server supports installation using installation sources provided with SLP and contains many system services with integrated support for SLP. You can use SLP to provide networked clients with central functions, such as an installation server, file server, or print server on your system. Services that offer SLP support include cupsd, login, ntp, openldap2, postfix, rpasswd, rsyncd, saned, sshd (via fish), vnc, and ypserv.

All packages necessary to use SLP services on a network client are installed by default. However, if you want to *provide* services via SLP, check that the `openslp-server` package is installed.

28.1 The SLP Front-End `slptool`

`slptool` is a command line tool to query and register SLP services. The query functions are useful for diagnostic purposes. The most important `slptool` subcommands are listed below. `slptool --help` lists all available options and functions.

`findsrvtypes`

List all service types available on the network.

```
tux > slptool findsrvtypes
service:install.suse:nfs
service:install.suse:ftp
service:install.suse:http
service:install.suse:smb
service:ssh
service:fish
service:YaST.installation.suse:vnc
service:smtp
service:domain
service:management-software.IBM:hardware-management-console
service:rsync
service:ntp
```



```
service:ypserv
```

findsrvs service type

List all servers providing service type

```
tux > slptool findsrvs service:ntp
service:ntp://ntp.example.com:123,57810
service:ntp://ntp2.example.com:123,57810
```

findattrs service type //host

List attributes for service type on host

```
tux > slptool findattrs service:ntp://ntp.example.com
(owner=tux),(email=tux@example.com)
```

register service type //host:port "(attribute=value),(attribute=value)"

Registers service type on host with an optional list of attributes

```
slptool register service:ntp://ntp.example.com:57810 \
"(owner=tux),(email=tux@example.com)"
```

deregister service type //host

De-registers service type on host

```
slptool deregister service:ntp://ntp.example.com
```

For more information run **slptool --help**.

28.2 Providing Services via SLP

To provide SLP services, the SLP daemon (slpd) must be running. Like most system services in SUSE Linux Enterprise Server, slpd is controlled by means of a separate start script. After the installation, the daemon is inactive by default. To activate it for the current session, run **sudo systemctl start slpd**. If slpd should be activated on system start-up, run **sudo systemctl enable slpd**.

Many applications in SUSE Linux Enterprise Server have integrated SLP support via the `libslp` library. If a service has not been compiled with SLP support, use one of the following methods to make it available via SLP:

Static Registration with `/etc/slp.reg.d`

Create a separate registration file for each new service. The following example registers a scanner service:

```
## Register a sane service on this system
## en means english language
## 65535 disables the timeout, so the service registration does
## not need refreshes
service:scanner.sane://$HOSTNAME:6566,en,65535
watch-port-tcp=6566
description=SANE scanner daemon
```

The most important line in this file is the *service URL*, which begins with `service:.` This contains the service type (`scanner.sane`) and the address under which the service is available on the server. `$HOSTNAME` is automatically replaced with the full host name. The name of the TCP port on which the relevant service can be found follows, separated by a colon. Then enter the language in which the service should appear and the duration of registration in seconds. These should be separated from the service URL by commas. Set the value for the duration of registration between `0` and `65535`. `0` prevents registration. `65535` removes all restrictions.

The registration file also contains the two variables `watch-port-tcp` and `description`. `watch-port-tcp` links the SLP service announcement to whether the relevant service is active by having `slpd` check the status of the service. The second variable contains a more precise description of the service that is displayed in suitable browsers.



Tip: YaST and SLP

Some services brokered by YaST, such as an installation server or YOU server, perform this registration automatically when you activate SLP in the module dialogs. YaST then creates registration files for these services.

Static Registration with `/etc/slp.reg`

The only difference between this method and the procedure with `/etc/slp.reg.d` is that all services are grouped within a central file.

Dynamic Registration with slptool

If a service needs to be registered dynamically without the need of configuration files, use the `slptool` command line utility. The same utility can also be used to de-register an existing service offering without restarting `slpd`. See *Section 28.1, “The SLP Front-End `slptool`”* for details.

28.2.1 Setting up an SLP Installation Server

Announcing the installation data via SLP within your network makes the network installation much easier, since the installation data such as IP address of the server or the path to the installation media are automatically required via SLP query. Refer to *Book “Deployment Guide”, Chapter 7 “Setting Up the Server Holding the Installation Sources”* for instructions.

28.3 For More Information

RFC 2608, 2609, 2610

RFC 2608 generally deals with the definition of SLP. RFC 2609 deals with the syntax of the service URLs used in greater detail and RFC 2610 deals with DHCP via SLP.

<http://www.openslp.org> ↗

The home page of the OpenSLP project.

[/usr/share/doc/packages/openslp](#)

This directory contains the documentation for SLP coming with the `openslp-server` package, including a `README.SUSE` containing the SUSE Linux Enterprise Server details, the RFCs, and two introductory HTML documents. Programmers who want to use the SLP functions will find more information in the *Programmers Guide* that is included in the `openslp-devel` package that is provided with the SUSE Software Development Kit.

29 The Apache HTTP Server

According to the survey from <http://www.netcraft.com/>, the Apache HTTP Server (Apache) is the world's most widely-used Web server. Developed by the Apache Software Foundation (<http://www.apache.org/>), it is available for most operating systems. SUSE® Linux Enterprise Server includes Apache version 2.4. In this chapter, learn how to install, configure and set up a Web server; how to use SSL, CGI, and additional modules; and how to troubleshoot Apache.

29.1 Quick Start

With this section, quickly set up and start Apache. You must be root to install and configure Apache.

29.1.1 Requirements

Make sure the following requirements are met before trying to set up the Apache Web server:

1. The machine's network is configured properly. For more information about this topic, refer to *Chapter 15, Basic Networking*.
2. The machine's exact system time is maintained by synchronizing with a time server. This is necessary because parts of the HTTP protocol depend on the correct time. See *Chapter 22, Time Synchronization with NTP* to learn more about this topic.
3. The latest security updates are installed. If in doubt, run a YaST Online Update.
4. The default Web server port (80) is opened in the firewall. For this, configure the SuSE-Firewall2 to allow the service *HTTP Server* in the external zone. This can be done using YaST. See Book “Security Guide”, *Chapter 15 “Masquerading and Firewalls”, Section 15.4.1 “Configuring the Firewall with YaST”* for details.

29.1.2 Installation

Apache on SUSE Linux Enterprise Server is not installed by default. To install it with a standard, predefined configuration that runs “out of the box”, proceed as follows:

PROCEDURE 29.1: INSTALLING APACHE WITH THE DEFAULT CONFIGURATION

1. Start YaST and select *Software > Software Management*.
2. Choose *View > Patterns* and select *Web and LAMP Server*.
3. Confirm the installation of the dependent packages to finish the installation process.

29.1.3 Start

You can start Apache automatically at boot time or start it manually.

To make sure that Apache is automatically started during boot in the targets `multi-user.target` and `graphical.target`, execute the following command:

```
root # systemctl enable apache2
```

For more information about the systemd targets in SUSE Linux Enterprise Server and a description of the YaST *Services Manager*, refer to [Section 13.4, “Managing Services with YaST”](#).

To manually start Apache using the shell, run `systemctl start apache2`.

PROCEDURE 29.2: CHECKING IF APACHE IS RUNNING

If you do not receive error messages when starting Apache, this usually indicates that the Web server is running. To test this:

1. Start a browser and open <http://localhost/>.
If Apache is up and running, you get a test page stating “It works!”.
2. If you do not see this page, refer to [Section 29.9, “Troubleshooting”](#).

Now that the Web server is running, you can add your own documents, adjust the configuration according to your needs, or add functionality by installing modules.

29.2 Configuring Apache

SUSE Linux Enterprise Server offers two configuration options:

- *Configuring Apache Manually*
- *Configuring Apache with YaST*

Manual configuration offers a higher level of detail, but lacks the convenience of the YaST GUI.



Important: Reload or Restart Apache after Configuration Changes

Most configuration changes require a reload (some also a restart) of Apache to take effect. Manually reload Apache with **`systemctl reload apache2`** or use one of the restart options as described in *Section 29.3, “Starting and Stopping Apache”*.

If you configure Apache with YaST, this can be taken care of automatically if you set *HTTP Service* to *Enabled* as described in *Section 29.2.3.2, “HTTP Server Configuration”*.

29.2.1 Apache Configuration Files

This section gives an overview of the Apache configuration files. If you use YaST for configuration, you do not need to touch these files—however, the information might be useful for you if you want to switch to manual configuration later on.

Apache configuration files can be found in two different locations:

- `/etc/sysconfig/apache2`
- `/etc/apache2/`

29.2.1.1 `/etc/sysconfig/apache2`

`/etc/sysconfig/apache2` controls some global settings of Apache, like modules to load, additional configuration files to include, flags with which the server should be started, and flags that should be added to the command line. Every configuration option in this file is extensively documented and therefore not mentioned here. For a general-purpose Web server, the settings in `/etc/sysconfig/apache2` should be sufficient for any configuration needs.

29.2.1.2 `/etc/apache2/`

`/etc/apache2/` hosts all configuration files for Apache. In the following, the purpose of each file is explained. Each file includes several configuration options (also called *directives*). Every configuration option in these files is extensively documented and therefore not mentioned here.

The Apache configuration files are organized as follows:

```
/etc/apache2/
|
| - charset.conv
| - conf.d/
|   |
|   | - *.conf
|
| - default-server.conf
| - errors.conf
| - httpd.conf
| - listen.conf
| - magic
| - mime.types
| - mod_*.conf
| - server-tuning.conf
| - ssl.*
| - ssl-global.conf
| - sysconfig.d
|   |
|   | - global.conf
|   | - include.conf
|   | - loadmodule.conf . .
|
| - uid.conf
| - vhosts.d
|   | - *.conf
```

APACHE CONFIGURATION FILES IN `/ETC/APACHE2/`

charset.conv

Specifies which character sets to use for different languages. Do not edit this file.

conf.d/*.conf

Configuration files added by other modules. These configuration files can be included into your virtual host configuration where needed. See `vhosts.d/vhost.template` for examples. By doing so, you can provide different module sets for different virtual hosts.

default-server.conf

Global configuration for all virtual hosts with reasonable defaults. Instead of changing the values, overwrite them with a virtual host configuration.

errors.conf

Defines how Apache responds to errors. To customize these messages for all virtual hosts, edit this file. Otherwise overwrite these directives in your virtual host configurations.

httpd.conf

The main Apache server configuration file. Avoid changing this file. It primarily contains include statements and global settings. Overwrite global settings in the pertinent configuration files listed here. Change host-specific settings (such as document root) in your virtual host configuration.

listen.conf

Binds Apache to specific IP addresses and ports. Name-based virtual hosting is also configured here. For details, see [Section 29.2.2.1.1, “Name-Based Virtual Hosts”](#).

magic

Data for the mime_magic module that helps Apache automatically determine the MIME type of an unknown file. Do not change this file.

mime.types

MIME types known by the system (this actually is a link to /etc/mime.types). Do not edit this file. If you need to add MIME types not listed here, add them to mod_mime-defaults.conf.

mod_*.conf

Configuration files for the modules that are installed by default. Refer to [Section 29.4, “Installing, Activating, and Configuring Modules”](#) for details. Note that configuration files for optional modules reside in the directory conf.d.

server-tuning.conf

Contains configuration directives for the different MPMs (see [Section 29.4.4, “Multiprocessing Modules”](#)) and general configuration options that control Apache's performance. Properly test your Web server when making changes here.

ssl-global.conf and ssl.*

Global SSL configuration and SSL certificate data. Refer to [Section 29.6, “Setting Up a Secure Web Server with SSL”](#) for details.

sysconfig.d/*.conf

Configuration files automatically generated from /etc/sysconfig/apache2. Do not change any of these files—edit /etc/sysconfig/apache2 instead. Do not put other configuration files in this directory.

uid.conf

Specifies under which user and group ID Apache runs. Do not change this file.

vhosts.d/*.conf

Your virtual host configuration should be located here. The directory contains template files for virtual hosts with and without SSL. Every file in this directory ending with .conf is automatically included in the Apache configuration. Refer to [Section 29.2.2.1, “Virtual Host Configuration”](#) for details.

29.2.2 Configuring Apache Manually

Configuring Apache manually involves editing plain text configuration files as user root.

29.2.2.1 Virtual Host Configuration

The term *virtual host* refers to Apache's ability to serve multiple universal resource identifiers (URIs) from the same physical machine. This means that several domains, such as `www.example.com` and `www.example.net`, are run by a single Web server on one physical machine.

It is common practice to use virtual hosts to save administrative effort (only a single Web server needs to be maintained) and hardware expenses (each domain does not require a dedicated server). Virtual hosts can be name based, IP based, or port based.

To list all existing virtual hosts, use the command `httpd2 -S`. This outputs a list showing the default server and all virtual hosts together with their IP addresses and listening ports. Furthermore, the list also contains an entry for each virtual host showing its location in the configuration files.

Virtual hosts can be configured via YaST as described in [Section 29.2.3.1.4, “Virtual Hosts”](#) or by manually editing a configuration file. By default, Apache in SUSE Linux Enterprise Server is prepared for one configuration file per virtual host in /etc/apache2/vhosts.d/. All files in


this directory with the extension `.conf` are automatically included to the configuration. A basic template for a virtual host is provided in this directory (`vhost.template` or `vhost-ssl.template` for a virtual host with SSL support).



Tip: Always Create a Virtual Host Configuration

It is recommended to always create a virtual host configuration file, even if your Web server only hosts one domain. By doing so, you not only have the domain-specific configuration in one file, but you can always fall back to a working basic configuration by simply moving, deleting, or renaming the configuration file for the virtual host. For the same reason, you should also create separate configuration files for each virtual host.

When using name-based virtual hosts it is recommended to set up a default configuration that will be used when a domain name does not match a virtual host configuration. The default virtual host is the one whose configuration is loaded first. Since the order of the configuration files is determined by file name, start the file name of the default virtual host configuration with an underscore character (`_`) to make sure it is loaded first (for example: `_default_vhost.conf`).

The `<VirtualHost>` `</VirtualHost>` block holds the information that applies to a particular domain. When Apache receives a client request for a defined virtual host, it uses the directives enclosed in this section. Almost all directives can be used in a virtual host context. See <http://httpd.apache.org/docs/2.4/mod/quickreference.html>  for further information about Apache's configuration directives.

29.2.2.1.1 Name-Based Virtual Hosts

With name-based virtual hosts, more than one Web site is served per IP address. Apache uses the host field in the HTTP header that is sent by the client to connect the request to a matching `ServerName` entry of one of the virtual host declarations. If no matching `ServerName` is found, the first specified virtual host is used as a default.

The first step is to create a `<VirtualHost>` block for each different name-based host that you want to serve. Inside each `<VirtualHost>` block, you will need at minimum a `ServerName` directive to designate which host is served and a `DocumentRoot` directive to show where in the file system the content for that host resides.

EXAMPLE 29.1: BASIC EXAMPLES OF NAME-BASED VirtualHost ENTRIES

```
<VirtualHost *:80>
# This first-listed virtual host is also the default for *:80
ServerName www.example.com
ServerAlias example.com
DocumentRoot /srv/www/htdocs/domain
</VirtualHost>

<VirtualHost *:80>
ServerName other.example.com
DocumentRoot /srv/www/htdocs/otherdomain
</VirtualHost>
```

The opening `VirtualHost` tag takes the IP address (or fully qualified domain name) as an argument in a name-based virtual host configuration. A port number directive is optional.

The wild card `*` is also allowed as a substitute for the IP address. When using IPv6 addresses, the address must be included in square brackets.

EXAMPLE 29.2: NAME-BASED VirtualHost DIRECTIVES

```
<VirtualHost 192.168.3.100:80>
...
</VirtualHost>

<VirtualHost 192.168.3.100>
...
</VirtualHost>

<VirtualHost *:80>
...
</VirtualHost>

<VirtualHost *>
...
</VirtualHost>

<VirtualHost [2002:c0a8:364::]>
...
</VirtualHost>
```

29.2.2.1.2 IP-Based Virtual Hosts

This alternative virtual host configuration requires the setup of multiple IPs for a machine. One instance of Apache hosts several domains, each of which is assigned a different IP.

The physical server must have one IP address for each IP-based virtual host. If the machine does not have multiple network cards, virtual network interfaces (IP aliasing) can also be used.

The following example shows Apache running on a machine with the IP 192.168.3.100, hosting two domains on the additional IPs 192.168.3.101 and 192.168.3.102. A separate VirtualHost block is needed for every virtual server.

EXAMPLE 29.3: IP-BASED VirtualHost DIRECTIVES

```
<VirtualHost 192.168.3.101>
...
</VirtualHost>

<VirtualHost 192.168.3.102>
...
</VirtualHost>
```

Here, VirtualHost directives are only specified for interfaces other than 192.168.3.100. When a Listen directive is also configured for 192.168.3.100, a separate IP-based virtual host must be created to answer HTTP requests to that interface—otherwise the directives found in the default server configuration (/etc/apache2/default-server.conf) are applied.

29.2.2.1.3 Basic Virtual Host Configuration

At least the following directives should be in each virtual host configuration to set up a virtual host. See /etc/apache2/vhosts.d/vhost.template for more options.

ServerName

The fully qualified domain name under which the host should be addressed.

DocumentRoot

Path to the directory from which Apache should serve files for this host. For security reasons, access to the entire file system is forbidden by default, so you must explicitly unlock this directory within a Directory container.

ServerAdmin

E-mail address of the server administrator. This address is, for example, shown on error pages Apache creates.

ErrorLog

The error log file for this virtual host. Although it is not necessary to create separate error log files for each virtual host, it is common practice to do so, because it makes the debugging of errors much easier. /var/log/apache2/ is the default directory for Apache's log files.

CustomLog

The access log file for this virtual host. Although it is not necessary to create separate access log files for each virtual host, it is common practice to do so, because it allows the separate analysis of access statistics for each host. /var/log/apache2/ is the default directory for Apache's log files.

As mentioned above, access to the whole file system is forbidden by default for security reasons. Therefore, explicitly unlock the directories in which you have placed the files Apache should serve—for example the DocumentRoot:

```
<Directory "/srv/www/www.example.com/htdocs">
    Require all granted
</Directory>
```



Note: Require all granted

In previous versions of Apache, the statement Require all granted was expressed as:

```
Order allow,deny
Allow from all
```

This old syntax is still supported by the mod_access_compat module.

The complete configuration file looks like this:

EXAMPLE 29.4: BASIC VirtualHost CONFIGURATION

```
<VirtualHost 192.168.3.100>
    ServerName www.example.com
    DocumentRoot /srv/www/www.example.com/htdocs
    ServerAdmin webmaster@example.com
    ErrorLog /var/log/apache2/www.example.com_log
    CustomLog /var/log/apache2/www.example.com-access_log common
    <Directory "/srv/www/www.example.com/htdocs">
        Require all granted
    </Directory>
```

29.2.3 Configuring Apache with YaST

To configure your Web server with YaST, start YaST and select *Network Services* › *HTTP Server*. When starting the module for the first time, the *HTTP Server Wizard* starts, prompting you to make a few basic decisions concerning administration of the server. After having finished the wizard, the *HTTP Server Configuration* dialog starts each time you call the *HTTP Server* module. For more information, see [Section 29.2.3.2, “HTTP Server Configuration”](#).

29.2.3.1 HTTP Server Wizard

The HTTP Server Wizard consists of five steps. In the last step of the dialog, you may enter the expert configuration mode to make even more specific settings.

29.2.3.1.1 Network Device Selection

Here, specify the network interfaces and ports Apache uses to listen for incoming requests. You can select any combination of existing network interfaces and their respective IP addresses. Ports from all three ranges (well-known ports, registered ports, and dynamic or private ports) that are not reserved by other services can be used. The default setting is to listen on all network interfaces (IP addresses) on port 80.

Check *Open Port In Firewall* to open the ports in the firewall that the Web server listens on. This is necessary to make the Web server available on the network, which can be a LAN, WAN, or the public Internet. Keeping the port closed is only useful in test situations where no external access to the Web server is necessary. If you have multiple network interfaces, click *Firewall Details* to specify on which interface(s) the port(s) should be opened.

Click *Next* to continue with the configuration.

29.2.3.1.2 Modules

The *Modules* configuration option allows for the activation or deactivation of the script languages that the Web server should support. For the activation or deactivation of other modules, refer to [Section 29.2.3.2.2, “Server Modules”](#). Click *Next* to advance to the next dialog.

29.2.3.1.3 Default Host

This option pertains to the default Web server. As explained in [Section 29.2.2.1, “Virtual Host Configuration”](#), Apache can serve multiple virtual hosts from a single physical machine. The first declared virtual host in the configuration file is commonly called the *default host*. Each virtual host inherits the default host's configuration.

To edit the host settings (also called *directives*), select the appropriate entry in the table then click *Edit*. To add new directives, click *Add*. To delete a directive, select it and click *Delete*.

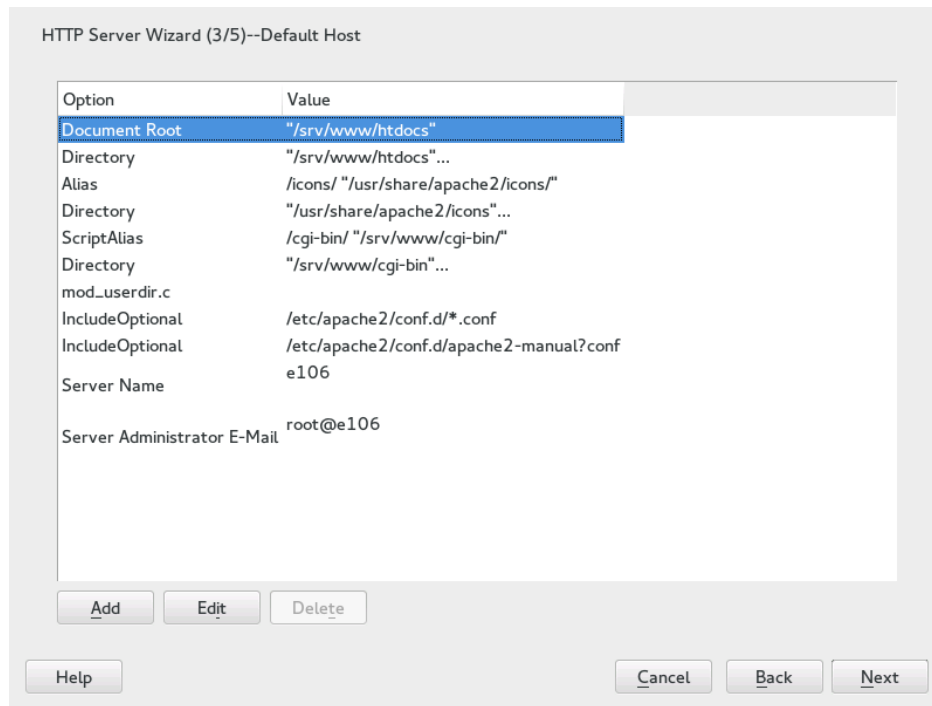


FIGURE 29.1: HTTP SERVER WIZARD: DEFAULT HOST

Here is list of the default settings of the server:

Document Root

Path to the directory from which Apache serves files for this host. `/srv/www/htdocs` is the default location.

Alias

With the help of Alias directives, URLs can be mapped to physical file system locations. This means that a certain path even outside the Document Root in the file system can be accessed via a URL aliasing that path.

The default SUSE Linux Enterprise Server Alias `/icons` points to `/usr/share/apache2/icons` for the Apache icons displayed in the directory index view.

ScriptAlias

Similar to the Alias directive, the ScriptAlias directive maps a URL to a file system location. The difference is that ScriptAlias designates the target directory as a CGI location, meaning that CGI scripts should be executed in that location.

Directory

With Directory settings, you can enclose a group of configuration options that will only apply to the specified directory.

Access and display options for the directories /srv/www/htdocs, /usr/share/apache2/icons and /srv/www/cgi-bin are configured here. It should not be necessary to change the defaults.

Include

With include, additional configuration files can be specified. Two Include directives are already preconfigured: /etc/apache2/conf.d/ is the directory containing the configuration files that come with external modules. With this directive, all files in this directory ending in .conf are included. With the second directive, /etc/apache2/conf.d/apache2-manual.conf, the apache2-manual configuration file is included.

Server Name

This specifies the default URL used by clients to contact the Web server. Use a fully qualified domain name (FQDN) to reach the Web server at http://FQDN/ or its IP address. You cannot choose an arbitrary name here—the server must be “known” under this name.

Server Administrator E-Mail

E-mail address of the server administrator. This address is, for example, shown on error pages Apache creates.

After finishing with the *Default Host* step, click *Next* to continue with the configuration.

29.2.3.1.4 Virtual Hosts

In this step, the wizard displays a list of already configured virtual hosts (see [Section 29.2.2.1, “Virtual Host Configuration”](#)). If you have not made manual changes prior to starting the YaST HTTP wizard, no virtual host is present.

To add a host, click *Add* to open a dialog in which to enter basic information about the host, such as *Server Name*, *Server Contents Root* (DocumentRoot), and the *Administrator E-Mail*. *Server Resolution* is used to determine how a host is identified (name based or IP based). Specify the name or IP address with *Change Virtual Host ID*

Clicking *Next* advances to the second part of the virtual host configuration dialog.

In part two of the virtual host configuration you can specify whether to enable CGI scripts and which directory to use for these scripts. It is also possible to enable SSL. If you do so, you must specify the path to the certificate as well. See [Section 29.6.2, “Configuring Apache with SSL”](#) for details on SSL and certificates. With the *Directory Index* option, you can specify which file to display when the client requests a directory (by default, `index.html`). Add one or more file names (space-separated) if you want to change this. With *Enable Public HTML*, the content of the users public directories (`~user/public_html/`) is made available on the server under <http://www.example.com/~user>.



Important: Creating Virtual Hosts

It is not possible to add virtual hosts at will. If using name-based virtual hosts, each host name must be resolved on the network. If using IP-based virtual hosts, you can assign only one host to each IP address available.

29.2.3.1.5 Summary

This is the final step of the wizard. Here, determine how and when the Apache server is started: when booting or manually. Also see a short summary of the configuration made so far. If you are satisfied with your settings, click *Finish* to complete configuration. If you want to change something, click *Back* until you have reached the desired dialog. Clicking *HTTP Server Expert Configuration* opens the dialog described in [Section 29.2.3.2, “HTTP Server Configuration”](#).

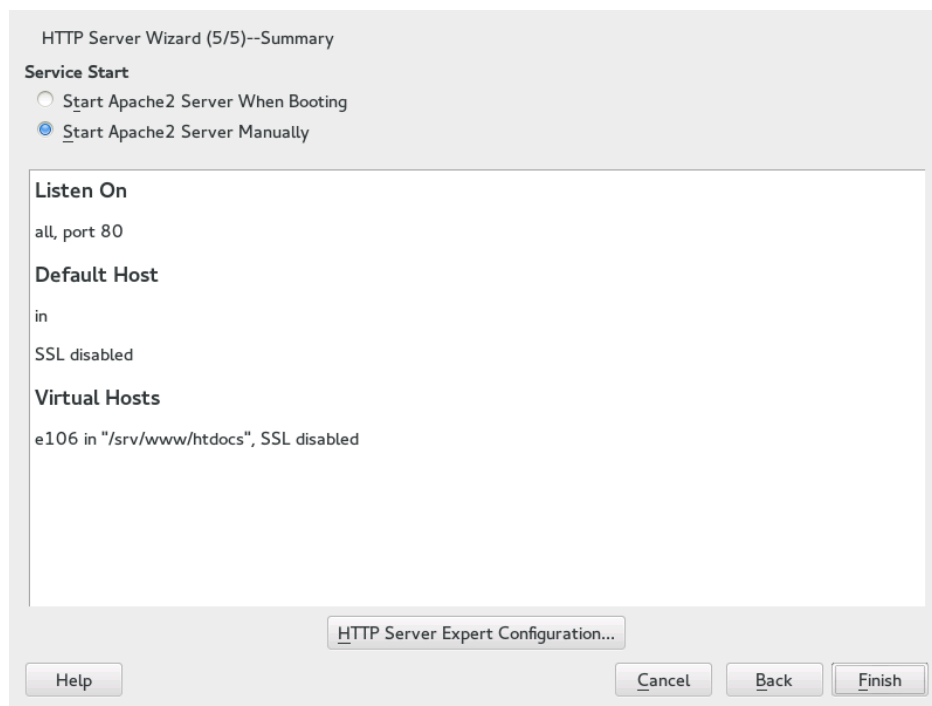


FIGURE 29.2: HTTP SERVER WIZARD: SUMMARY

29.2.3.2 HTTP Server Configuration

The *HTTP Server Configuration* dialog also lets you make even more adjustments to the configuration than the wizard (which only runs if you configure your Web server for the first time). It consists of four tabs described in the following. No configuration option you change here is effective immediately—you always must confirm your changes with *Finish* to make them effective. Clicking *Abort* leaves the configuration module and discards your changes.

29.2.3.2.1 Listen Ports and Addresses

In *HTTP Service*, select whether Apache should be running (*Enabled*) or stopped (*Disabled*). In *Listen on Ports*, *Add*, *Edit*, or *Delete* addresses and ports on which the server should be available. The default is to listen on all interfaces on port 80. You should always check *Open Port In Firewall*, because otherwise the Web server is not reachable from outside. Keeping the port closed is only useful in test situations where no external access to the Web server is necessary. If you have multiple network interfaces, click *Firewall Details* to specify on which interface(s) the port(s) should be opened.

With *Log Files*, watch either the access log file or the error log file. This is useful if you want to test your configuration. The log file opens in a separate window from which you can also restart or reload the Web server. For details, see [Section 29.3, “Starting and Stopping Apache”](#). These commands are effective immediately and their log messages are also displayed immediately.

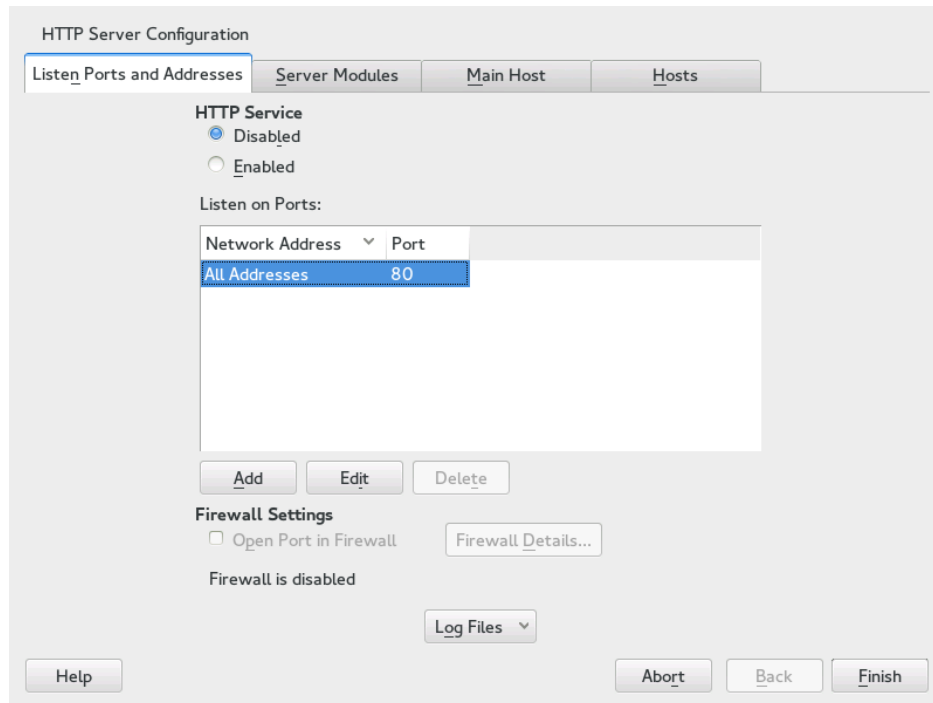


FIGURE 29.3: HTTP SERVER CONFIGURATION: LISTEN PORTS AND ADDRESSES

29.2.3.2.2 Server Modules

You can change the status (enabled or disabled) of Apache2 modules by clicking *Toggle Status*. Click *Add Module* to add a new module that is already installed but not yet listed. Learn more about modules in [Section 29.4, “Installing, Activating, and Configuring Modules”](#).

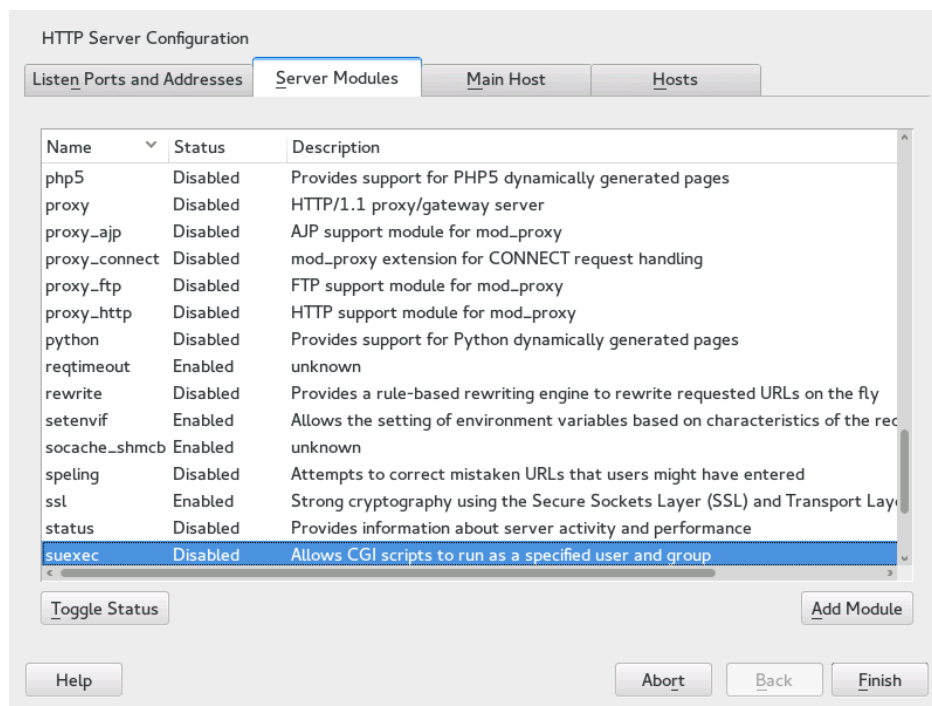


FIGURE 29.4: HTTP SERVER CONFIGURATION: SERVER MODULES

29.2.3.2.3 Main Host or Hosts

These dialogs are identical to the ones already described. Refer to [Section 29.2.3.1.3, “Default Host”](#) and [Section 29.2.3.1.4, “Virtual Hosts”](#).

29.3 Starting and Stopping Apache

If configured with YaST as described in [Section 29.2.3, “Configuring Apache with YaST”](#), Apache is started at boot time in the `multi-user.target` and `graphical.target`. You can change this behavior using YaST's *Services Manager* or with the `systemctl` command line tool (`systemctl enable` or `systemctl disable`).

To start, stop, or manipulate Apache on a running system, use either the `systemctl` or the `apachectl` commands as described below.

For general information about `systemctl` commands, refer to [Section 13.2.1, “Managing Services in a Running System”](#).

`systemctl status apache2`

Checks if Apache is started.

systemctl start apache2

Starts Apache if it is not already running.

systemctl stop apache2

Stops Apache by terminating the parent process.

systemctl restart apache2

Stops and then restarts Apache. Starts the Web server if it was not running before.

systemctl try-restart apache2

Stops then restarts Apache only if it is already running.

systemctl reload apache2

Stops the Web server by advising all forked Apache processes to first finish their requests before shutting down. As each process dies, it is replaced by a newly started one, resulting in a complete “restart” of Apache.

apachectl -k graceful

Starts a second Web server that immediately serves all incoming requests. The previous instance of the Web server continues to handle all existing requests for a defined period of time configured with GracefulShutdownTimeout.

This command is useful either when upgrading to a new version or when having changed configuration options that require a restart. Using this option ensures a minimum server downtime.

If GracefulShutdownTimeout is set to zero, the server will wait indefinitely until all remaining requests have been fully served.

A graceful restart can fail if the original Apache instance is not able to clear all necessary resources. In this case, the command will result in a graceful stop.

systemctl stop apache2

Stops the Web server after a defined period of time configured with GracefulShutdownTimeout to ensure that existing requests can be finished.

apachectl configtest

Checks the syntax of the configuration files without affecting a running Web server. Because this check is forced every time the server is started, reloaded, or restarted, it is usually not necessary to run the test explicitly (if a configuration error is found, the Web server is not started, reloaded, or restarted).



Tip: Additional Flags

If you specify additional flags to the commands, these are passed through to the Web server.

29.4 Installing, Activating, and Configuring Modules

The Apache software is built in a modular fashion: all functionality except some core tasks are handled by modules. This has progressed so far that even HTTP is processed by a module (`http_core`).

Apache modules can be compiled into the Apache binary at build time or dynamically loaded at runtime. Refer to [Section 29.4.2, “Activation and Deactivation”](#) for details of how to load modules dynamically.

Apache modules can be divided into four different categories:

Base Modules

Base modules are compiled into Apache by default. Apache in SUSE Linux Enterprise Server has only `mod_so` (needed to load other modules) and `http_core` compiled in. All others are available as shared objects: rather than being included in the server binary itself, they can be included at runtime.

Extension Modules

In general, modules labeled as extensions are included in the Apache software package, but are usually not compiled into the server statically. In SUSE Linux Enterprise Server, they are available as shared objects that can be loaded into Apache at runtime.

External Modules

Modules labeled external are not included in the official Apache distribution. However, SUSE Linux Enterprise Server provides several of them.

Multiprocessing Modules (MPMs)

MPMs are responsible for accepting and handling requests to the Web server, representing the core of the Web server software.

29.4.1 Module Installation

If you have done a default installation as described in [Section 29.1.2, “Installation”](#), the following modules are already installed: all base and extension modules, the multiprocessing module `Pre-fork MPM`, and the external module `mod_python`.

You can install additional external modules by starting YaST and choosing *Software > Software Management*. Now choose *View > Search* and search for *apache*. Among other packages, the results list contains all available external Apache modules.

29.4.2 Activation and Deactivation

Activate or deactivate particular modules either manually or with YaST. In YaST, script language modules (PHP5, Perl, and Python) need to be enabled or disabled with the module configuration described in [Section 29.2.3.1, “HTTP Server Wizard”](#). All other modules can be enabled or disabled as described in [Section 29.2.3.2.2, “Server Modules”](#).

If you prefer to activate or deactivate the modules manually, use the commands `a2enmod mod_foo` or `a2dismod mod_foo`, respectively. `a2enmod -l` outputs a list of all currently active modules.



Important: Including Configuration Files for External Modules

If you have activated external modules manually, make sure to load their configuration files in all virtual host configurations. Configuration files for external modules are located under `/etc/apache2/conf.d/` and are loaded in `/etc/apache2/default-server.conf` by default. For more fine-grained control you can comment out the inclusion in `/etc/apache2/default-server.conf` and add it to specific virtual hosts only. See `/etc/apache2/vhosts.d/vhost.template` for examples.

29.4.3 Base and Extension Modules

All base and extension modules are described in detail in the Apache documentation. Only a brief description of the most important modules is available here. Refer to <http://httpd.apache.org/docs/2.4/mod/> to learn details about each module.

mod_actions

Provides methods to execute a script whenever a certain MIME type (such as application/pdf), a file with a specific extension (like .rpm), or a certain request method (such as GET) is requested. This module is enabled by default.


mod_alias

Provides Alias and Redirect directives with which you can map a URL to a specific directory (Alias) or redirect a requested URL to another location. This module is enabled by default.

mod_auth*

The authentication modules provide different authentication methods: basic authentication with mod_auth_basic or digest authentication with mod_auth_digest.

mod_auth_basic and mod_auth_digest must be combined with an authentication provider module, mod_authn_* (for example, mod_authn_file for text file-based authentication) and with an authorization module mod_authz_* (for example, mod_authz_user for user authorization).

More information about this topic is available in the *Authentication HOWTO* at <http://httpd.apache.org/docs/2.4/howto/auth.html> .

mod_autoindex

Autoindex generates directory listings when no index file (for example, index.html) is present. The look and feel of these indexes is configurable. This module is enabled by default. However, directory listings are disabled by default via the Options directive—overwrite this setting in your virtual host configuration. The default configuration file for this module is located at /etc/apache2/mod_autoindex-defaults.conf.

mod_cgi

mod_cgi is needed to execute CGI scripts. This module is enabled by default.

mod_deflate

Using this module, Apache can be configured to compress given file types on the fly before delivering them.

mod_dir

mod_dir provides the DirectoryIndex directive with which you can configure which files are automatically delivered when a directory is requested (index.html by default). It also provides an automatic redirect to the correct URL when a directory request does not contain a trailing slash. This module is enabled by default.

mod_env

Controls the environment that is passed to CGI scripts or SSI pages. Environment variables can be set or unset or passed from the shell that invoked the httpd process. This module is enabled by default.

mod_expires

With mod_expires, you can control how often proxy and browser caches refresh your documents by sending an Expires header. This module is enabled by default.

mod_http2

With mod_http2, Apache gains support for the HTTP/2 protocol. It can be enabled by specifying Protocols h2 http/1.1 in a VirtualHost.

mod_include

mod_include lets you use Server Side Includes (SSI), which provide a basic functionality to generate HTML pages dynamically. This module is enabled by default.

mod_info

Provides a comprehensive overview of the server configuration under `http://localhost/server-info/`. For security reasons, you should always limit access to this URL. By default only `localhost` is allowed to access this URL. mod_info is configured at /etc/apache2/mod_info.conf.


mod_log_config

With this module, you can configure the look of the Apache log files. This module is enabled by default.

mod_mime

The mime module makes certain that a file is delivered with the correct MIME header based on the file name's extension (for example text/html for HTML documents). This module is enabled by default.

mod_negotiation

Necessary for content negotiation. See <http://httpd.apache.org/docs/2.4/content-negotiation.html>  for more information. This module is enabled by default.

mod_rewrite

Provides the functionality of mod_alias, but offers more features and flexibility. With mod_rewrite, you can redirect URLs based on multiple rules, request headers, and more.

mod_setenvif

Sets environment variables based on details of the client's request, such as the browser string the client sends, or the client's IP address. This module is enabled by default.

mod_spelling

mod_spelling attempts to automatically correct typographical errors in URLs, such as capitalization errors.

mod_ssl

Enables encrypted connections between Web server and clients. See [Section 29.6, “Setting Up a Secure Web Server with SSL”](#) for details. This module is enabled by default.

mod_status

Provides information on server activity and performance under `http://localhost/server-status/`. For security reasons, you should always limit access to this URL. By default, only localhost is allowed to access this URL. mod_status is configured at /etc/apache2/mod_status.conf.

mod_suexec

mod_suexec lets you run CGI scripts under a different user and group. This module is enabled by default.

mod_userdir

Enables user-specific directories available under ~user/. The UserDir directive must be specified in the configuration. This module is enabled by default.

29.4.4 Multiprocessing Modules

SUSE Linux Enterprise Server provides two different multiprocessing modules (MPMs) for use with Apache:

- *Prefork MPM*
- *Worker MPM*

29.4.4.1 Prefork MPM

The prefork MPM implements a non-threaded, preforking Web server. It makes the Web server behave similarly to Apache version 1.x. In this version it isolates each request and handles it by forking a separate child process. Thus problematic requests cannot affect others, avoiding a lockup of the Web server.

While providing stability with this process-based approach, the prefork MPM consumes more system resources than its counterpart, the worker MPM. The prefork MPM is considered the default MPM for Unix-based operating systems.



Important: MPMs in This Document

This document assumes Apache is used with the prefork MPM.

29.4.4.2 Worker MPM

The worker MPM provides a multi-threaded Web server. A thread is a “lighter” form of a process. The advantage of a thread over a process is its lower resource consumption. Instead of only forking child processes, the worker MPM serves requests by using threads with server processes. The preforked child processes are multi-threaded. This approach makes Apache perform better by consuming fewer system resources than the prefork MPM.

One major disadvantage is the stability of the worker MPM: if a thread becomes corrupt, all threads of a process can be affected. In the worst case, this may result in a server crash. Especially when using the Common Gateway Interface (CGI) with Apache under heavy load, internal server errors might occur because of threads being unable to communicate with system resources. Another argument against using the worker MPM with Apache is that not all available Apache modules are thread-safe and thus cannot be used with the worker MPM.



Warning: Using PHP Modules with MPMs

Not all available PHP modules are thread-safe. Using the worker MPM with `mod_php` is strongly discouraged.

29.4.5 External Modules

Find a list of all external modules shipped with SUSE Linux Enterprise Server here. Find the module's documentation in the listed directory.

mod_apparmor

Adds support to Apache to provide AppArmor confinement to individual CGI scripts handled by modules like mod_php5 and mod_perl.

Package Name: apache2-mod_apparmor

More Information: *Book* “Security Guide”

mod_perl

mod_perl enables you to run Perl scripts in an embedded interpreter. The persistent interpreter embedded in the server avoids the overhead of starting an external interpreter and the penalty of Perl start-up time.

Package Name: apache2-mod_perl

Configuration File: /etc/apache2/conf.d/mod_perl.conf

More Information: /usr/share/doc/packages/apache2-mod_perl

mod_php5

PHP is a server-side, cross-platform HTML embedded scripting language.

Package Name: apache2-mod_php5

Configuration File: /etc/apache2/conf.d/php5.conf

More Information: /usr/share/doc/packages/apache2-mod_php5

mod_python

mod_python allows embedding Python within the Apache HTTP server for a considerable boost in performance and added flexibility in designing Web-based applications.

Package Name: apache2-mod_python

More Information: /usr/share/doc/packages/apache2-mod_python

mod_security

mod_security provides a Web application firewall to protect Web applications from a range of attacks. It also enables HTTP traffic monitoring and real-time analysis.

Package Name: apache2-mod_security2

Configuration File: /etc/apache2/conf.d/mod_security2.conf

More Information: /usr/share/doc/packages/apache2-mod_security2

Documentation: <http://modsecurity.org/documentation/> 

29.4.6 Compilation

Apache can be extended by advanced users by writing custom modules. To develop modules for Apache or compile third-party modules, the package `apache2-devel` is required along with the corresponding development tools. `apache2-devel` also contains the `apxs2` tools, which are necessary for compiling additional modules for Apache.

`apxs2` enables the compilation and installation of modules from source code (including the required changes to the configuration files), which creates *dynamic shared objects* (DSOs) that can be loaded into Apache at runtime.

The `apxs2` binaries are located under `/usr/sbin`:

- `/usr/sbin/apxs2`—suitable for building an extension module that works with any MPM. The installation location is `/usr/lib64/apache2`.
- `/usr/sbin/apxs2-prefork`—suitable for prefork MPM modules. The installation location is `/usr/lib64/apache2-prefork`.
- `/usr/sbin/apxs2-worker`—suitable for worker MPM modules. The installation location is `/usr/lib64/apache2-worker`.

Install and activate a module from source code with the following commands:

```
cd /path/to/module/source
apxs2 -c -i -a mod_foo.c
```

where `-c` compiles the module, `-i` installs it, and `-a` activates it. Other options of `apxs2` are described in the `apxs2(1)` man page.

29.5 Enabling CGI Scripts

Apache's Common Gateway Interface (CGI) lets you create dynamic content with programs or scripts usually called CGI scripts. CGI scripts can be written in any programming language. Usually, script languages such as Perl or PHP are used.

To enable Apache to deliver content created by CGI scripts, `mod_cgi` needs to be activated. `mod_alias` is also needed. Both modules are enabled by default. Refer to [Section 29.4.2, “Activation and Deactivation”](#) for details on activating modules.



Warning: CGI Security

Allowing the server to execute CGI scripts is a potential security hole. Refer to [Section 29.8, “Avoiding Security Problems”](#) for additional information.

29.5.1 Apache Configuration

In SUSE Linux Enterprise Server, the execution of CGI scripts is only allowed in the directory `/srv/www/cgi-bin/`. This location is already configured to execute CGI scripts. If you have created a virtual host configuration (see [Section 29.2.2.1, “Virtual Host Configuration”](#)) and want to place your scripts in a host-specific directory, you must unlock and configure this directory.

EXAMPLE 29.5: VIRTUALHOST CGI CONFIGURATION

```
ScriptAlias /cgi-bin/ "/srv/www/www.example.com/cgi-bin/" ❶

<Directory "/srv/www/www.example.com/cgi-bin/">
  Options +ExecCGI ❷
  AddHandler cgi-script .cgi .pl ❸
  Require all granted ❹
</Directory>
```

- ❶ Tells Apache to handle all files within this directory as CGI scripts.
- ❷ Enables CGI script execution
- ❸ Tells the server to treat files with the extensions `.pl` and `.cgi` as CGI scripts. Adjust according to your needs.
- ❹ The `Require` directive controls the default access state. In this case, access is granted to the specified directory without limitation. For more information on authentication and authorization, see <http://httpd.apache.org/docs/2.4/howto/auth.html> ❷.

29.5.2 Running an Example Script

CGI programming differs from "regular" programming in that the CGI programs and scripts must be preceded by a MIME-Type header such as `Content-type: text/html`. This header is sent to the client, so it understands what kind of content it receives. Secondly, the script's output must be something the client, usually a Web browser, understands—HTML usually, or plain text or images, for example.

A simple test script available under `/usr/share/doc/packages/apache2/test-cgi` is part of the Apache package. It outputs the content of some environment variables as plain text. Copy this script to either `/srv/www/cgi-bin/` or the script directory of your virtual host (`/srv/www/www.example.com/cgi-bin/`) and name it `test.cgi`. Edit the file to have `#!/bin/sh` as the first line.

Files accessible by the Web server should be owned by the user `root`. For additional information see [Section 29.8, "Avoiding Security Problems"](#). Because the Web server runs with a different user, the CGI scripts must be world-executable and world-readable. Change into the CGI directory and use the command `chmod 755 test.cgi` to apply the proper permissions.

Now call `http://localhost/cgi-bin/test.cgi` or `http://www.example.com/cgi-bin/test.cgi`. You should see the "CGI/1.0 test script report".

29.5.3 CGI Troubleshooting

If you do not see the output of the test program but an error message instead, check the following:

CGI TROUBLESHOOTING

- If you have configured your custom CGI directory, is it configured properly? If in doubt, try the script within the default CGI directory `/srv/www/cgi-bin/` and call it with `http://localhost/cgi-bin/test.cgi`.
- Are the file permissions correct? Change into the CGI directory and execute `ls -l test.cgi`. Its output should start with

```
-rwxr-xr-x 1 root root
```

- Make sure that the script does not contain programming errors. If you have not changed `test.cgi`, this should not be the case, but if you are using your own programs, always make sure that they do not contain programming errors.

29.6 Setting Up a Secure Web Server with SSL

Whenever sensitive data, such as credit card information, is transferred between Web server and client, it is desirable to have a secure, encrypted connection with authentication. `mod_ssl` provides strong encryption using the secure sockets layer (SSL) and transport layer security (TLS) protocols for HTTP communication between a client and the Web server. Using SSL/TLS, a private connection between Web server and client is established. Data integrity is ensured and client and server can authenticate each other.

For this purpose, the server sends an SSL certificate that holds information proving the server's valid identity before any request to a URL is answered. In turn, this guarantees that the server is the uniquely correct end point for the communication. Additionally, the certificate generates an encrypted connection between client and server that can transport information without the risk of exposing sensitive, plain-text content.

`mod_ssl` does not implement the SSL/TLS protocols itself, but acts as an interface between Apache and an SSL library. In SUSE Linux Enterprise Server, the OpenSSL library is used. OpenSSL is automatically installed with Apache.

The most visible effect of using `mod_ssl` with Apache is that URLs are prefixed with `https://` instead of `http://`.

29.6.1 Creating an SSL Certificate

To use SSL/TLS with the Web server, you need to create an SSL certificate. This certificate is needed for the authorization between Web server and client, so that each party can clearly identify the other party. To ensure the integrity of the certificate, it must be signed by a party every user trusts.

There are three types of certificates you can create: a “dummy” certificate for testing purposes only, a self-signed certificate for a defined circle of users that trust you, and a certificate signed by an independent, publicly-known certificate authority (CA).

Creating a certificate is a two step process. First, a private key for the certificate authority is generated then the server certificate is signed with this key.



Tip: For More Information

To learn more about concepts and definitions of SSL/TLS, refer to http://httpd.apache.org/docs/2.4/ssl/ssl_intro.html.

29.6.1.1 Creating a “Dummy” Certificate

To generate a dummy certificate, call the script `/usr/bin/gensslcert`. It creates or overwrites the files listed below. Use `gensslcert`'s optional switches to fine-tune the certificate. Call `/usr/bin/gensslcert -h` for more information.

- `/etc/apache2/ssl.crt/ca.crt`
- `/etc/apache2/ssl.crt/server.crt`
- `/etc/apache2/ssl.key/server.key`
- `/etc/apache2/ssl.csr/server.csr`

A copy of `ca.crt` is also placed at `/srv/www/htdocs/CA.crt` for download.



Important: For Testing Purposes Only

A dummy certificate should never be used on a production system. Only use it for testing purposes.

29.6.1.2 Creating a Self-Signed Certificate

If you are setting up a secure Web server for an intranet or for a defined circle of users, it is probably sufficient if you sign a certificate with your own certificate authority (CA). Note that the visitors to such a site will see the annoying "this is an untrusted site" warning because Web browsers do not know the self-signed certificate.



Important: Self-Signed Certificates

Only use a self-signed certificate on a Web server that is accessed by people who know and trust you as a certificate authority. It is not recommended to use such a certificate for a public shop, for example.

First you need to generate a certificate signing request (CSR). You are going to use **openssl**, with **PEM** as the certificate format. During this step, you will be asked for a passphrase, and to answer several questions. Remember the passphrase you enter as you will need it in the future.

```
sudo openssl req -new > new.cert.csr
Generating a 1024 bit RSA private key
.....+++++
writing new private key to 'privkey.pem'
Enter PEM pass phrase: ❶
Verifying - Enter PEM pass phrase: ❷
-----
You are about to be asked to enter information that will be incorporated
into your certificate request.
What you are about to enter is what is called a Distinguished Name or a DN.
There are quite a few fields but you can leave some blank
For some fields there will be a default value,
If you enter '.', the field will be left blank.
-----
Country Name (2 letter code) [AU]: ❸
State or Province Name (full name) [Some-State]: ❹
Locality Name (eg, city) []: ❺
Organization Name (eg, company) [Internet Widgits Pty Ltd]: ❻
Organizational Unit Name (eg, section) []: ❼
Common Name (for example server FQDN, or YOUR name) []: ❽
Email Address []: ❾

Please enter the following 'extra' attributes
to be sent with your certificate request
A challenge password []: ❿
An optional company name []: ⓫
```

- ❶ Fill in your passphrase,
- ❷ ...fill it in once more (and remember it).
- ❸ Fill in your 2 letter country code, such as GB or CZ.
- ❹ Fill in the name of the state where you live.
- ❺ Fill in the city name, such as Prague.
- ❻ Fill in the name of the organization you work for.
- ❼ Fill in your organization unit, or leave blank if you have none.
- ❽ Fill in either the domain name of the server, or your first and last name.

- 9 Fill in your work e-mail address.
- 10 Leave the challenge password empty, otherwise you will need to enter it every time you restart the Apache Web server.
- 11 Fill in the optional company name, or leave blank.

Now you can generate the certificate. You are going to use **openssl** again, and the format of the certificate is the default **PEM**.

PROCEDURE 29.3: GENERATING THE CERTIFICATE

1. Export the private part of the key to new.cert.key. You will be prompted for the passphrase you entered when creating the certificate signing request (CSR).

```
sudo openssl rsa -in privkey.pem -out new.cert.key
```

2. Generate the public part of the certificate according to the information you filled out in the signing request. The -days option specifies the length of time before the certificate expires. You can revoke a certificate, or replace one before it expires.

```
sudo openssl x509 -in new.cert.csr -out new.cert.cert -req \
-signkey new.cert.key -days 365
```

3. Copy the certificate files to the relevant directories, so that the Apache server can read them. Make sure that the private key /etc/apache2/ssl.key/server.key is not world-readable, while the public PEM certificate /etc/apache2/ssl.crt/server.crt is.


```
sudo cp new.cert.cert /etc/apache2/ssl.crt/server.crt
sudo cp new.cert.key /etc/apache2/ssl.key/server.key
```



Tip: Public Certificate Location

The last step is to copy the public certificate file from /etc/apache2/ssl.crt/server.crt to a location where your users can access it to incorporate it into the list of known and trusted CAs in their Web browsers. Otherwise a browser complains that the certificate was issued by an unknown authority.

29.6.1.3 Getting an Officially Signed Certificate

There are several official certificate authorities that sign your certificates. The certificate is signed by a trustworthy third party, so can be fully trusted. Publicly operating secure Web servers usually have an officially signed certificate. A list of the most used Certificate Authorities (CAs) is available at https://en.wikipedia.org/wiki/Certificate_authority#Providers .

When requesting an officially signed certificate, you do not send a certificate to the CA. Instead, issue a Certificate Signing Request (CSR). To create a CSR, run the following command:

```
openssl req -new -newkey rsa:2048 -nodes -keyout newkey.pem -out newreq.pem
```

You are asked to enter a distinguished name. This requires you to answer a few questions, such as country name or organization name. Enter valid data—everything you enter here later shows up in the certificate and is checked. You do not need to answer every question. If one does not apply to you or you want to leave it blank, use “.”. Common name is the name of the CA itself—choose a significant name, such as My company CA. Last, a challenge password and an alternative company name must be entered.

Find the CSR in the directory from which you called the script. The file is named newreq.pem.

29.6.2 Configuring Apache with SSL

The default port for SSL and TLS requests on the Web server side is 443. There is no conflict between a “regular” Apache listening on port 80 and an SSL/TLS-enabled Apache listening on port 443. In fact, HTTP and HTTPS can be run with the same Apache instance. Usually separate virtual hosts are used to dispatch requests to port 80 and port 443 to separate virtual servers.



Important: Firewall Configuration

Do not forget to open the firewall for SSL-enabled Apache on port 443. This can be done with YaST as described in *Book “Security Guide”, Chapter 15 “Masquerading and Firewalls”, Section 15.4.1 “Configuring the Firewall with YaST”*.

The SSL module is enabled by default in the global server configuration. In case it has been disabled on your host, activate it with the following command: **a2enmod ssl**. To finally enable SSL, the server needs to be started with the flag “SSL”. To do so, call **a2enflag SSL** (case-sensitive!). If you have chosen to encrypt your server certificate with a password, you should

also increase the value for `APACHE_TIMEOUT` in `/etc/sysconfig/apache2`, so you have enough time to enter the passphrase when Apache starts. Restart the server to make these changes active. A reload is not sufficient.

The virtual host configuration directory contains a template `/etc/apache2/vhosts.d/vhost-ssl.template` with SSL-specific directives that are extensively documented. Refer to [Section 29.2.2.1, “Virtual Host Configuration”](#) for the general virtual host configuration.

To get started, copy the template to `/etc/apache2/vhosts.d/mySSL-host.conf` and edit it. Adjusting the values for the following directives should be sufficient:

- DocumentRoot
- ServerName
- ServerAdmin
- ErrorLog
- TransferLog

29.6.2.1 Name-Based Virtual Hosts and SSL

By default it is not possible to run multiple SSL-enabled virtual hosts on a server with only one IP address. Name-based virtual hosting requires that Apache knows which server name has been requested. The problem with SSL connections is, that such a request can only be read after the SSL connection has already been established (by using the default virtual host). As a result, users will receive a warning message stating that the certificate does not match the server name.

SUSE Linux Enterprise Server comes with an extension to the SSL protocol called Server Name Indication (SNI) addresses this issue by sending the name of the virtual domain as part of the SSL negotiation. This enables the server to “switch” to the correct virtual domain early and present the browser the correct certificate.

SNI is enabled by default on SUSE Linux Enterprise Server. To enable Name-Based Virtual Hosts for SSL, configure the server as described in [Section 29.2.2.1.1, “Name-Based Virtual Hosts”](#) (note that you need to use port `443` rather than port `80` with SSL).

! Important: SNI Browser Support

SNI must also be supported on the client side. Although SNI is supported by most browsers, some browsers for mobile hardware as well as Internet Explorer and Safari on Windows* XP lack SNI support. See http://en.wikipedia.org/wiki/Server_Name_Indication for details.

Configure how to handle non-SNI capable browser with the directive `SSLStrictSNIVHostCheck`. When set to `on` in the server configuration, non-SNI capable browser will be rejected for all virtual hosts. When set to `on` within a `VirtualHost` directive, access to this particular Host will be rejected.

When set to `off` in the server configuration, the server will behave as if not having SNI support. SSL requests will be handled by the *first* Virtual host defined (for port 443).

29.7 Running Multiple Apache Instances on the Same Server

As of SUSE® Linux Enterprise Server 12 SP1, you can run multiple Apache instances on the same server. This has several advantages over running multiple virtual hosts (see [Section 29.2.2.1, “Virtual Host Configuration”](#)):

- When a virtual host needs to be disabled for some time, you need to change the Web server configuration and restart it so that the change takes effect.
- In case of problems with one virtual host, you need to restart all of them.

You can run the default Apache instance as usual:

```
systemctl start apache2
```

It reads the default `/etc/sysconfig/apache2` file. If the file is not present, or it is present but it does not set the `APACHE_HTTPD_CONF` variable, it reads `/etc/apache2/httpd.conf`.

To activate another Apache instance, run:

```
systemctl start apache2@instance_name
```

For example:

```
systemctl start apache2@example_web.org
```

By default, the instance uses `/etc/apache2@example_web.org/httpd.conf` as a main configuration file, which can be overwritten by setting `APACHE_HTTPD_CONF` in `/etc/sysconfig/apache2@example_web.org`.

An example to set up an additional instance of Apache follows. Note that you need to execute all the commands as `root`.

PROCEDURE 29.4: CONFIGURING AN ADDITIONAL APACHE INSTANCE

1. Create a new configuration file based on `/etc/sysconfig/apache2`, for example `/etc/sysconfig/apache2@example_web.org`:

```
cp /etc/sysconfig/apache2 /etc/sysconfig/apache2@example_web.org
```

2. Edit the file `/etc/sysconfig/apache2@example_web.org` and change the line containing

```
APACHE_HTTPD_CONF
```

to

```
APACHE_HTTPD_CONF="/etc/apache2/httpd@example_web.org.conf"
```

3. Create the file `/etc/apache2/httpd@example_web.org.conf` based on `/etc/apache2/httpd.conf`.

```
cp /etc/apache2/httpd.conf /etc/apache2/httpd@example_web.org.conf
```

4. Edit `/etc/apache2/httpd@example_web.org.conf` and change

```
Include /etc/apache2/listen.conf
```

to

```
Include /etc/apache2/listen@example_web.org.conf
```

Review all the directives and change them to fit your needs. You will probably want to change

```
Include /etc/apache2/global.conf
```

and create new `global@example_web.org.conf` for each instance. We suggest to change

```
ErrorLog /var/log/apache2/error_log
```

to

```
ErrorLog /var/log/apache2/error@example_web.org_log
```

to have separate logs for each instance.

5. Create `/etc/apache2/listen@example_web.org.conf` based on `/etc/apache2/listen.conf`.

```
cp /etc/apache2/listen.conf /etc/apache2/listen@example_web.org.conf
```

6. Edit `/etc/apache2/listen@example_web.org.conf` and change

```
Listen 80
```

to the port number you want the new instance to run on, for example 82:

```
Listen 82
```

If you want to run the new Apache instance over a secured protocol (see [Section 29.6, “Setting Up a Secure Web Server with SSL”](#)), change also the line

```
Listen 443
```

for example to

```
Listen 445
```

7. Start the new Apache instance:

```
systemctl start apache2@example_web.org
```


8. Check if the server is running by pointing your Web browser at `http://server_name:82`. If you previously changed the name of the error log file for the new instance, you can check it:

```
tail -f /var/log/apache2/error@example_web.org_log
```

Here are several points to consider when setting up more Apache instances on the same server:

- The file `/etc/sysconfig/apache2@instance_name` can include the same variables as `/etc/sysconfig/apache2`, including module loading and MPM setting.
- The default Apache instance does not need to be running while other instances run.
- The Apache helper utilities `a2enmod`, `a2dismod` and `apachectl` operate on the default Apache instance if not specified otherwise with the `HTTPD_INSTANCE` environment variable. The following example

```
export HTTPD_INSTANCE=example_web.org
a2enmod access_compat
a2enmod status
apachectl start
```

will add `access_compat` and `status` modules to the `APACHE_MODULES` variable of `/etc/sysconfig/apache2@example_web.org`, and then start the `example_web.org` instance.

29.8 Avoiding Security Problems

A Web server exposed to the public Internet requires an ongoing administrative effort. It is inevitable that security issues appear, both related to the software and to accidental misconfiguration. Here are some tips for how to deal with them.

29.8.1 Up-to-Date Software

If there are vulnerabilities found in the Apache software, a security advisory will be issued by SUSE. It contains instructions for fixing the vulnerabilities, which in turn should be applied when possible. The SUSE security announcements are available from the following locations:

- **Web Page.** <http://www.suse.com/support/security/> ↗
- **Mailing List Archive.** <http://lists.opensuse.org/opensuse-security-announce/> ↗
- **List of Security Announcements.** <http://www.suse.com/support/update/> ↗

29.8.2 DocumentRoot Permissions

By default in SUSE Linux Enterprise Server, the DocumentRoot directory /srv/www/htdocs and the CGI directory /srv/www/cgi-bin belong to the user and group root. You should not change these permissions. If the directories are writable for all, any user can place files into them. These files might then be executed by Apache with the permissions of wwwrun, which may give the user unintended access to file system resources. Use subdirectories of /srv/www to place the DocumentRoot and CGI directories for your virtual hosts and make sure that directories and files belong to user and group root.

29.8.3 File System Access

By default, access to the whole file system is denied in /etc/apache2/httpd.conf. You should never overwrite these directives, but specifically enable access to all directories Apache should be able to read. For details, see *Section 29.2.2.1.3, “Basic Virtual Host Configuration”*. In doing so, ensure that no critical files, such as password or system configuration files, can be read from the outside.

29.8.4 CGI Scripts

Interactive scripts in Perl, PHP, SSI, or any other programming language can essentially run arbitrary commands and therefore present a general security issue. Scripts that will be executed from the server should only be installed from sources the server administrator trusts—allowing users to run their own scripts is generally not a good idea. It is also recommended to do security audits for all scripts.

To make the administration of scripts as easy as possible, it is common practice to limit the execution of CGI scripts to specific directories instead of globally allowing them. The directives `ScriptAlias` and `Option ExecCGI` are used for configuration. The SUSE Linux Enterprise Server default configuration does not allow execution of CGI scripts from everywhere.

All CGI scripts run as the same user, so different scripts can potentially conflict with each other. The module `suEXEC` lets you run CGI scripts under a different user and group.

29.8.5 User Directories

When enabling user directories (with `mod_userdir` or `mod_rewrite`) you should strongly consider not allowing `.htaccess` files, which would allow users to overwrite security settings. At least you should limit the user's engagement by using the directive `AllowOverride`. In SUSE Linux Enterprise Server, `.htaccess` files are enabled by default, but the user is not allowed to overwrite any `Option` directives when using `mod_userdir` (see the `/etc/apache2/mod_userdir.conf` configuration file).

29.9 Troubleshooting

If Apache does not start, the Web page is not accessible, or users cannot connect to the Web server, it is important to find the cause of the problem. Here are some typical places to look for error explanations and important things to check:

Output of the `apache2.service` subcommand:

Instead of starting and stopping the Web server with the binary `/usr/sbin/httpd2`, rather use the **`systemctl`** commands instead (described in [Section 29.3, “Starting and Stopping Apache”](#)). It is verbose about errors, and it even provides tips and hints for fixing configuration errors.

Log Files and Verbosity

In case of both fatal and nonfatal errors, check the Apache log files for causes, mainly the error log file located at `/var/log/apache2/error_log` by default. Additionally, you can control the verbosity of the logged messages with the `LogLevel` directive if more detail is needed in the log files.



Tip: A Simple Test

Watch the Apache log messages with the command `tail -F /var/log/apache2/my_error_log`. Then run `systemctl restart apache2`. Now, try to connect with a browser and check the output.

Firewall and Ports

A common mistake is to not open the ports for Apache in the firewall configuration of the server. If you configure Apache with YaST, there is a separate option available to take care of this specific issue (see [Section 29.2.3, “Configuring Apache with YaST”](#)). If you are configuring Apache manually, open firewall ports for HTTP and HTTPS via YaST's firewall module.

If the error cannot be tracked down with any of these, check the online Apache bug database at http://httpd.apache.org/bug_report.html. Additionally, the Apache user community can be reached via a mailing list available at <http://httpd.apache.org/userslist.html>.

29.10 For More Information

The package `apache2-doc` contains the complete Apache manual in various localizations for local installation and reference. It is not installed by default—the quickest way to install it is to use the command `zypper in apache2-doc`. Having been installed, the Apache manual is available at <http://localhost/manual/>. You may also access it on the Web at <http://httpd.apache.org/docs-2.4/>. SUSE-specific configuration hints are available in the directory `/usr/share/doc/packages/apache2/README.*`.

29.10.1 Apache 2.4

For a list of new features in Apache 2.4, refer to http://httpd.apache.org/docs/2.4/new_features_2_4.html. Information about upgrading from version 2.2 to 2.4 is available at <http://httpd.apache.org/docs-2.4/upgrading.html>.

29.10.2 Apache Modules

More information about external Apache modules that are briefly described in *Section 29.4.5, “External Modules”* is available at the following locations:

mod_apparmor

<http://en.opensuse.org/SDB:AppArmor> ↗

mod_auth_kerb

<http://modauthkerb.sourceforge.net/> ↗

mod_perl

<http://perl.apache.org/> ↗

mod_php5

<http://www.php.net/manual/en/install.unix.apache2.php> ↗

mod_python

<http://www.modpython.org/> ↗

mod_security

<http://modsecurity.org/> ↗

29.10.3 Development

More information about developing Apache modules or about getting involved in the Apache Web server project are available at the following locations:

Apache Developer Information

<http://httpd.apache.org/dev/> ↗

Apache Developer Documentation

<http://httpd.apache.org/docs/2.4/developer/> ↗

Writing Apache Modules with Perl and C

<http://www.modperl.com/> ↗

29.10.4 Miscellaneous Sources

If you experience difficulties specific to Apache in SUSE Linux Enterprise Server, take a look at the Technical Information Search at <http://www.suse.com/support> . The history of Apache is provided at http://httpd.apache.org/ABOUT_APACHE.html . This page also explains why the server is called Apache.

30 Setting Up an FTP Server with YaST

Using the YaST *FTP Server* module, you can configure your machine to function as an FTP (File Transfer Protocol) server. Anonymous and/or authenticated users can connect to your machine and download files using the FTP protocol. Depending on the configuration, they can also upload files to the FTP server. YaST uses vsftpd (Very Secure FTP Daemon).

If the YaST FTP Server module is not available in your system, install the `yast2-ftp-server` package.

To configure the FTP server using YaST, follow these steps:

1. Open the YaST control center and choose *Network Services > FTP Server* or run the `yast2-ftp-server` command as `root`.
2. If there is not any FTP server installed in your system, you will be asked which server to install when the YaST FTP Server module starts. Choose the vsftpd server and confirm the dialog.
3. In the *Start-Up* dialog, configure the options for starting of the FTP server. For more information, see [Section 30.1, "Starting the FTP Server"](#).
In the *General* dialog, configure FTP directories, welcome message, file creation masks and various other parameters. For more information, see [Section 30.2, "FTP General Settings"](#).
In the *Performance* dialog, set the parameters that affect the load on the FTP server. For more information, see [Section 30.3, "FTP Performance Settings"](#).
In the *Authentication* dialog, set whether the FTP server should be available for anonymous and/or authenticated users. For more information, see [Section 30.4, "Authentication"](#).
In the *Expert Settings* dialog, configure the operation mode of the FTP server, SSL connections and firewall settings. For more information, see [Section 30.5, "Expert Settings"](#).
4. Press *Finish* to save the configurations.

30.1 Starting the FTP Server

In the *Service Start* frame of the *FTP Start-Up* dialog set the way the FTP server is started up. You can choose between starting the server automatically during the system boot and starting it manually. If the FTP server should be started only after an FTP connection request, choose *Via xinetd*.

The current status of the FTP server is shown in the *Switch On and Off* frame of the *FTP Start-Up* dialog. Start the FTP server by clicking *Start FTP Now*. To stop the server, click *Stop FTP Now*. After having changed the settings of the server click *Save Settings and Restart FTP Now*. Your configurations will be saved by leaving the configuration module with *Finish*.

The *Selected Service* frame of the *FTP Start-Up* dialog shows which FTP server is used: either vsftpd or pure-ftpd. If both servers are installed, you can switch between them—the current configuration will automatically be converted.

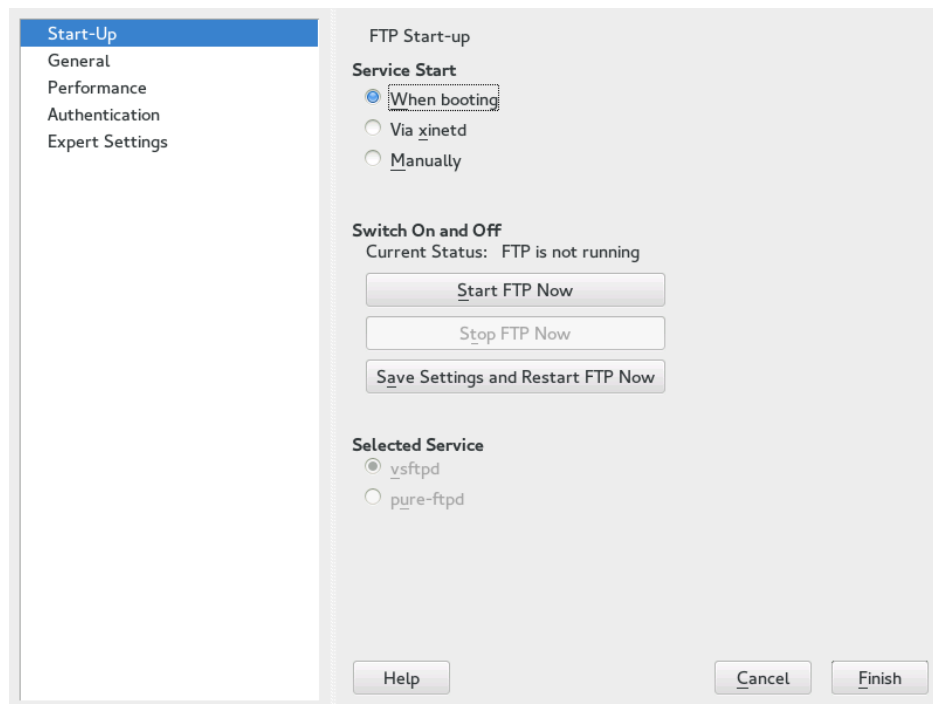


FIGURE 30.1: FTP SERVER CONFIGURATION — START-UP

30.2 FTP General Settings

In the *General Settings* frame of the *FTP General Settings* dialog you can set the *Welcome message* which is shown after connecting to the FTP server.

If you check the *Chroot Everyone* option, all local users will be placed in a chroot jail in their home directory after login. This option has security implications, especially if the users have upload permission or shell access, so be careful enabling this option.

If you check the *Verbose Logging* option, all FTP requests and responses are logged.

You can limit permissions of files created by anonymous and/or authenticated users with `umask`. Set the file creation mask for anonymous users in *Umask for Anonymous* and the file creation mask for authenticated users in *Umask for Authenticated Users*. The masks should be entered as octal numbers with a leading zero. For more information about `umask`, see the `umask` man page (`man 1p umask`).

In the *FTP Directories* frame set the directories used for anonymous and authorized users. With *Browse*, you can select a directory to be used from the local file system. The default FTP directory for anonymous users is `/srv/ftp`. Note that `vsftpd` does not allow this directory to be writable for all users. The subdirectory `upload` with write permissions for anonymous users is created instead.



Note: Write Permissions in FTP Directory

The `pure-ftpd` server allows the FTP directory for anonymous users to be writable. When switching between servers, make sure you remove the write permissions in the directory that was used with `pure-ftpd` before switching back to the `vsftpd` server.

30.3 FTP Performance Settings

In the *Performance* dialog set the parameters which affect the load on the FTP server. *Max Idle Time* is the maximum time (in minutes) the remote client may spend between FTP commands. In case of longer inactivity, the remote client is disconnected. *Max Clients for One IP* determines the maximum number of clients which can be connected from a single IP address. *Max Clients* determines the maximum number of clients which may be connected. Any additional clients will get an error message.

The maximum data transfer rate (in KB/s) is set in *Local Max Rate* for local authenticated users, and in *Anonymous Max Rate* for anonymous clients respectively. The default value for the rate settings is `0`, which means unlimited data transfer rate.

30.4 Authentication

In the *Enable/Disable Anonymous and Local Users* frame of the *Authentication* dialog, you can set which users are allowed to access your FTP server. You can choose between the following options: granting access to anonymous users only, to authenticated users only (with accounts on the system) or to both types of users.

If you want to allow users to upload files to the FTP server, check *Enable Upload* in the *Uploading* frame of the *Authentication* dialog. Here you are able to allow uploading or creating directories even for anonymous users by checking the respective box.



Note: vsftpd—Allowing File Upload for Anonymous Users

If a vsftpd server is used and you want anonymous users to be able to upload files or create directories, a subdirectory with writing permissions for all users needs to be created in the anonymous FTP directory.

30.5 Expert Settings

An FTP server can run in active or in passive mode. By default the server runs in passive mode. To switch into active mode, uncheck *Enable Passive Mode* option in *Expert Settings* dialog. You can also change the range of ports on the server used for the data stream by tweaking the *Min Port for Pas. Mode* and *Max Port for Pas. Mode* options.

If you want encrypted communication between clients and the server, you can *Enable SSL*. Check the versions of the protocol to be supported and specify the DSA certificate to be used for SSL encrypted connections.

If your system is protected by a firewall, check *Open Port in Firewall* to enable a connection to the FTP server.

30.6 For More Information

For more information about the FTP server read the manual pages of [`vsftpd`](#) and [`vsftpd.conf`](#).

31 The Proxy Server Squid

Squid is a widely-used proxy cache for Linux and Unix platforms. This means that it stores requested Internet objects, such as data on a Web or FTP server, on a machine that is closer to the requesting workstation than the server. It can be set up in multiple hierarchies to assure optimal response times and low bandwidth usage, even in modes that are transparent to end users. Additional software like squidGuard can be used to filter Web content.

Squid acts as a proxy cache. It redirects object requests from clients (in this case, from Web browsers) to the server. When the requested objects arrive from the server, it delivers the objects to the client and keeps a copy of them in the hard disk cache. An advantage of caching is that several clients requesting the same object can be served from the hard disk cache. This enables clients to receive the data much faster than from the Internet. This procedure also reduces the network traffic.

Along with actual caching, Squid offers a wide range of features:

- Distributing load over intercommunicating hierarchies of proxy servers
- Defining strict access control lists for all clients accessing the proxy
- Allowing or denying access to specific Web pages using other applications
- Generating statistics about frequently-visited Web pages for the assessment of surfing habits

Squid is not a generic proxy. It normally proxies only HTTP connections. It supports the protocols FTP, Gopher, SSL, and WAIS, but it does not support other Internet protocols, such as the news protocol, or video conferencing protocols. Because Squid only supports the UDP protocol to provide communication between different caches, many multimedia programs are not supported.

31.1 Some Facts about Proxy Caches

As a proxy cache, Squid can be used in several ways. When combined with a firewall, it can help with security. Multiple proxies can be used together. It can also determine what types of objects should be cached and for how long.

31.1.1 Squid and Security

It is possible to use Squid together with a firewall to secure internal networks from the outside using a proxy cache. The firewall denies all clients access to external services except Squid. All Web connections must be established by the proxy. With this configuration, Squid completely controls Web access.

If the firewall configuration includes a DMZ, the proxy should operate within this zone. [Section 31.5, “Configuring a Transparent Proxy”](#) describes how to implement a *transparent* proxy. This simplifies the configuration of the clients, because in this case, they do not need any information about the proxy.

31.1.2 Multiple Caches

Several instances of Squid can be configured to exchange objects between them. This reduces the total system load and increases the chances of retrieving an object from the local network. It is also possible to configure cache hierarchies, so a cache can forward object requests to sibling caches or to a parent cache—causing it to request objects from another cache in the local network or directly from the source.

Choosing the appropriate topology for the cache hierarchy is very important, because it is not desirable to increase the overall traffic on the network. For a very large network, it would make sense to configure a proxy server for every subnet and connect them to a parent proxy, which in turn is connected to the proxy cache of the ISP.

All this communication is handled by ICP (Internet cache protocol) running on top of the UDP protocol. Data transfers between caches are handled using HTTP (hypertext transmission protocol) based on TCP.

To find the most appropriate server from which to request objects, a cache sends an ICP request to all sibling proxies. The sibling proxies answer these requests via ICP responses. If the object was detected, they use the code HIT, if not, they use MISS.

If multiple HIT responses were found, the proxy server decides from which server to download, depending on factors such as which cache sent the fastest answer or which one is closer. If no satisfactory responses are received, the request is sent to the parent cache.



Note: How Squid Avoids Duplication of Objects

To avoid duplication of objects in different caches in the network, other ICP protocols are used, such as CARP (cache array routing protocol) or HTCP (hypertext cache protocol). The more objects maintained in the network, the greater the possibility of finding the desired one.

31.1.3 Caching Internet Objects

Many objects available in the network are not static, such as dynamically generated pages and TLS/SSL-encrypted content. Objects like these are not cached because they change each time they are accessed.

To determine how long objects should remain in the cache, objects are assigned one of several states. Web and proxy servers find out the status of an object by adding headers to these objects, such as “Last modified” or “Expires” and the corresponding date. Other headers specifying that objects must not be cached can be used as well.

Objects in the cache are normally replaced, because of a lack of free disk space, using algorithms such as LRU (last recently used). This means that the proxy expunges those objects that have not been requested for the longest time.

31.2 System Requirements

System requirements largely depend on the maximum network load that the system must bear. Therefore, examine load peaks, as during those times, load might be more than four times the day's average. When in doubt, slightly overestimate the system's requirements. Having Squid working close to the limit of its capabilities can lead to a severe loss in quality of service. The following sections point to system factors in order of significance:

1. RAM size
2. CPU speed/physical CPU cores
3. Size of the disk cache
4. Hard disks/SSDs and their architecture

31.2.1 RAM

The amount of memory (RAM) required by Squid directly correlates with the number of objects in the cache. Random access memory is much faster than a hard disk/SSD. Therefore, it is very important to have sufficient memory for the Squid process, because system performance is dramatically reduced if it must be swapped to disk.

Squid also stores cache object references and frequently requested objects in the main memory to speed up retrieval of this data. In addition to that, there is other data that Squid needs to keep in memory, such as a table with all the IP addresses handled, an exact domain name cache, the most frequently requested objects, access control lists, buffers, and more.

31.2.2 CPU

Squid is tuned to work best with lower processor core counts (4–8 physical cores), with each providing high performance. Technologies providing virtual cores such as hyperthreading can hurt performance.

To make the best use of multiple CPU cores, it is necessary to set up multiple worker threads writing to different caching devices. By default, multi-core support is mostly disabled.

31.2.3 Size of the Disk Cache

In a small cache, the probability of a HIT (finding the requested object already located there) is small, because the cache is easily filled and less requested objects are replaced by newer ones. If, for example, 1 GB is available for the cache and the users use up only 10 MB per day surfing, it would take more than one hundred days to fill the cache.

The easiest way to determine the necessary cache size is to consider the maximum transfer rate of the connection. With a 1 Mbit/s connection, the maximum transfer rate is 128 KB/s. If all this traffic ended up in the cache, in one hour it would add up to 460 MB. Assuming that all this traffic is generated in only eight working hours, it would reach 3.6 GB in one day. Because the connection is normally not used to its upper volume limit, it can be assumed that the total data volume handled by the cache is approximately 2 GB. Hence, in this example, 2 GB of disk space is required for Squid to keep one day's worth of browsing data cached.

31.2.4 Hard Disk/SSD Architecture

Speed plays an important role in the caching process, so this factor deserves special attention. For hard disks/SSDs, this parameter is described as *random seek time* or *random read performance*, measured in milliseconds. Because the data blocks that Squid reads from or writes to the hard disk/SSD tend to be small, the seek time/read performance of the hard disk/SSD is more important than its data throughput.

For use as a proxy, hard disks with high rotation speeds or SSDs are the best choice. When using hard disks, it can be better to use multiple smaller hard disks, each with a single cache directory to avoid excessive read times.

Using a RAID system allows increasing reliability at expense of speed. However, for performance reasons, avoid (software) RAID5 and similar settings.

File system choice is usually not decisive. However, using the mount option `noatime` can improve performance—Squid provides its own time stamps and thus does not need the file system to track access times.

31.3 Basic Usage of Squid

If not already installed, install the package `squid`. `squid` is not among the packages installed by default on SUSE® Linux Enterprise Server.

Squid is already preconfigured in SUSE Linux Enterprise Server, you can start it directly after the installation. To ensure a smooth start-up, the network should be configured in a way that at least one name server and the Internet can be reached. Problems can arise if a dial-up connection is used with a dynamic DNS configuration. In this case, at least the name server should be specified, because Squid does not start if it does not detect a DNS server in `/etc/resolv.conf`.

31.3.1 Starting Squid

To start Squid, use:

```
tux > sudo systemctl start squid
```

If you want Squid to start together with the system, enable the service with `systemctl enable squid`.

31.3.2 Checking Whether Squid Is Working

To check whether Squid is running, choose one of the following ways:

- Using **systemctl**:

```
tux > systemctl start squid
```

The output of this command should indicate that Squid is loaded and active (running).

- Using Squid itself:

```
tux > sudo squid -k | echo $?
```

The output of this command should be 0, without further messages.

To test the functionality of Squid on the local system, choose one of the following ways:

- To test, you can use **squidclient**, a command-line tool that can output the response to a Web request, similar to **wget** or **curl**.

Unlike those tools, **squidclient** will automatically connect to the default proxy setup of Squid, localhost:3128. However, if you changed the configuration of Squid, you need to configure **squidclient** to use different settings using command line options. For more information, see **squidclient --help**.

EXAMPLE 31.1: A REQUEST WITH squidclient

```
tux > squidclient http://www.example.org
HTTP/1.1 200 OK
Cache-Control: max-age=604800
Content-Type: text/html
Date: Fri, 22 Jun 2016 12:00:00 GMT
Expires: Fri, 29 Jun 2016 12:00:00 GMT
Last-Modified: Fri, 09 Aug 2013 23:54:35 GMT
Server: ECS (iad/182A)
Vary: Accept-Encoding
X-Cache: HIT
x-ec-custom-error: 1
Content-Length: 1270
X-Cache: MISS from moon❶
X-Cache-Lookup: MISS from moon:3128
Via: 1.1 moon (squid/3.5.16)❷
```



```
Connection: close

<!doctype html>
<html>
<head>
  <title>Example Domain</title>
[...]
```

The output shown in *Example 31.1, “A Request With **squidclient**”* can be split into two parts:

1. The protocol headers of the response: the lines before the blank line.
2. The actual content of the response: the lines after the blank line.

To verify that Squid is used, refer to the selected header lines:

- ❶ The value of the header X-Cache tells you that the requested document was not in the Squid cache (MISS) of the computer moon.
The example above contains two X-Cache lines. You can ignore the first X-Cache header. It is produced by the internal caching software of the originating Web server.
 - ❷ The value of the header Via tells you the HTTP version, the name of the computer, and the version of Squid in use.
- Using a browser: Set up localhost as the proxy and 3128 as the port. You can then load a page and check the response headers in the *Network* panel of the browser's *Inspector* or *Developer Tools*. The headers should be reproduced similarly to the way shown in *Example 31.1, “A Request With **squidclient**”*.

To allow users from the local system and other systems to access Squid and the Internet, change the entry in the configuration files /etc/squid/squid.conf from http_access deny all to http_access allow all. However, in doing so, consider that Squid is made completely accessible to anyone by this action. Therefore, define ACLs (access control lists) that control access to the proxy. After modifying the configuration file, Squid must be reloaded or restarted. For more information on ACLs, see *Section 31.4.2, “Options for Access Controls”*.

If Squid dies after a short period of time even though it was started successfully, check whether there is a faulty name server entry or whether the /etc/resolv.conf file is missing. Squid logs the cause of a start-up failure in the file /var/log/squid/cache.log.

31.3.3 Stopping, Reloading, and Restarting Squid

Do this with `systemctl reload squid`. Alternatively, completely restart Squid with `systemctl restart squid`.

The command `systemctl stop squid` causes Squid to shut down. This can take a while, because Squid waits up to half a minute (`shutdown_lifetime` option in `/etc/squid/squid.conf`) before dropping the connections to the clients and writing its data to the disk.



Warning: Terminating Squid

Terminating Squid with `kill` or `killall` can damage the cache. To be able to restart Squid, damaged caches must be deleted.

31.3.4 Removing Squid

Removing Squid from the system does not remove the cache hierarchy and log files. To remove these, delete the `/var/cache/squid` directory manually.

31.3.5 Local DNS Server

Setting up a local DNS server makes sense even if it does not manage its own domain. It then simply acts as a caching-only name server and is also able to resolve DNS requests via the root name servers without requiring any special configuration (see [Section 23.4, "Starting the BIND Name Server"](#)). How this can be done depends on whether you chose dynamic DNS during the configuration of the Internet connection.

Dynamic DNS

Normally, with dynamic DNS, the DNS server is set by the provider during the establishment of the Internet connection and the local `/etc/resolv.conf` file is adjusted automatically. This behavior is controlled in the `/etc/sysconfig/network/config` file with the `NETCONFIG_DNS_POLICY` sysconfig variable. Set `NETCONFIG_DNS_POLICY` to `""` with the YaST sysconfig editor.

Then, add the local DNS server in the `/etc/resolv.conf` file with the IP address `127.0.0.1` for `localhost`. This way, Squid can always find the local name server when it starts.

To make the provider's name server accessible, specify it in the configuration file `/etc/named.conf` under `forwarders` along with its IP address. With dynamic DNS, this can be achieved automatically when establishing the connection by setting the sysconfig variable `NETCONFIG_DNS_POLICY` to `auto`.

Static DNS

With static DNS, no automatic DNS adjustments take place while establishing a connection, so there is no need to change any sysconfig variables. However, you must specify the local DNS server in the file `/etc/resolv.conf` as described in *Dynamic DNS*. Additionally, the provider's static name server must be specified manually in the `/etc/named.conf` file under `forwarders` along with its IP address.



Tip: DNS and Firewall

If you have a firewall running, make sure DNS requests can pass it.

31.4 The `/etc/squid/squid.conf` Configuration File

All Squid proxy server settings are made in the `/etc/squid/squid.conf` file. To start Squid for the first time, no changes are necessary in this file, but external clients are initially denied access. The proxy is available for `localhost`. The default port is `3128`. The preinstalled configuration file `/etc/squid/squid.conf` provides detailed information about the options and many examples.

Many entries are commented and therefore begin with the comment character `#`. The relevant specifications can be found at the end of the line. The given values usually correlate with the default values, so removing the comment signs without changing any of the parameters usually has no effect. If possible, leave the commented lines as they are and insert the options along with the modified values in the line below. This way, the default values may easily be recovered and compared with the changes.



Tip: Adapting the Configuration File After an Update

If you have updated from an earlier Squid version, it is recommended to edit the new `/etc/squid/squid.conf` and only apply the changes made in the previous file.

Sometimes, Squid options are added, removed, or modified. Therefore, if you try to use the old squid.conf, Squid might stop working properly.

31.4.1 General Configuration Options

The following is a list of a selection of configuration options for Squid. It is not exhaustive. The Squid package contains a full, lightly documented list of options in /etc/squid/squid.conf.documented.

http_port PORT

This is the port on which Squid listens for client requests. The default port is 3128, but 8080 is also common.

cache_peer HOST_NAME TYPE PROXY_PORT ICP_PORT

This option allows creating a network of caches that work together. The cache peer is a computer that also hosts a network cache and stands in a relationship to your own. The type of relationship is specified as the TYPE. The type can either be parent or sibling. As the HOST_NAME, specify the name or IP address of the proxy to use. For PROXY_PORT, specify the port number for use in a browser (usually 8080). Set ICP_PORT to 7 or, if the ICP port of the parent is not known and its use is irrelevant to the provider, to 0.

To make Squid behave like a Web browser instead of like a proxy, prohibit the use of the ICP protocol. You can do so by appending the options default and no-query.

cache_mem SIZE

This option defines the amount of memory Squid can use for very popular replies. The default is 8 MB. This does not specify the memory usage of Squid and may be exceeded.

cache_dir STORAGE_TYPE CACHE_DIRECTORY CACHE_SIZE LEVEL_1_DIRECTORIES LEVEL_2_DIRECTORIES

The option cache_dir defines the directory for the disk cache. In the default configuration on SUSE Linux Enterprise Server, Squid does not create a disk cache.

The placeholder STORAGE_TYPE can be one of the following:

- Directory-based storage types: ufs, aufs (the default), diskd. All three are variations of the storage format ufs. However, while ufs runs as part of the core Squid thread, aufs runs in a separate thread, and diskd uses a separate process. This means that the latter two types avoid blocking Squid because of disk I/O.
- Database-based storage systems: rock. This storage format relies on a single database file, in which each object takes up one or more memory units of a fixed size (“slots”).

In the following, only the parameters for storage types based on ufs will be discussed. rock has somewhat different parameters.

The CACHE_DIRECTORY is the directory for the disk cache. By default, that is /var/cache/squid. CACHE_SIZE is the maximum size of that directory in megabytes; by default, this is set to 100 MB. Set it to between 50% and a maximum of 80% of available disk space.

The final two values, LEVEL_1_DIRECTORIES and LEVEL_2_DIRECTORIES specify how many subdirectories are created in the CACHE_DIRECTORY. By default, 16 subdirectories are created at the first level below CACHE_DIRECTORY and 256 within each of these. These values should only be increased with caution, because creating too many directories can lead to performance problems.

If you have several disks that share a cache, specify several cache_dir lines.

cache_access_log LOG_FILE ,
cache_log LOG_FILE ,
cache_store_log LOG_FILE

These three options specify the paths where Squid logs all its actions. Normally, nothing needs to be changed here. If Squid is burdened by heavy usage, it might make sense to distribute the cache and the log files over several disks.

client_netmask NETMASK

This option allows masking IP addresses of clients in the log files by applying a subnet mask. For example, to set the last digit of the IP address to 0, specify 255.255.255.0.

ftp_user E-MAIL

This option allows setting the password that Squid should use for anonymous FTP login. Specify a valid e-mail address here, because some FTP servers check these for validity.

cache_mgr E-MAIL

If it unexpectedly crashes, Squid sends a message to this e-mail address. The default is *webmaster*.

logfile_rotate VALUE

If you run **squid -k rotate**, **Squid** can rotate log files. The files are numbered in this process and, after reaching the specified value, the oldest file is overwritten. The default value is 10 which rotates log files with the numbers 0 to 9.

However, on SUSE Linux Enterprise Server, rotating log files is performed automatically using logrotate and the configuration file /etc/logrotate.d/squid.

append_domain DOMAIN

Use *append_domain* to specify which domain to append automatically when none is given. Usually, your own domain is specified here, so specifying *www* in the browser accesses your own Web server.

forwarded_for STATE

If this option is set to on, it adds a line to the header similar to this:

```
X-Forwarded-For: 192.168.0.1
```

If you set this option to off, Squid removes the IP address and the system name of the client from HTTP requests.

negative_ttl TIME ,

negative_dns_ttl TIME

If these options are set, Squid will cache some types of failures, such as 404 responses. It will then refuse to issue new requests, even if the resource would be available then.

By default, negative_ttl is set to 0, negative_dns_ttl is set to 1 minutes. This means that negative responses to Web requests are not cached by default, while negative responses to DNS requests are cached for 1 minute.

never_direct allow ACL_NAME

To prevent Squid from taking requests directly from the Internet, use the option never_direct to force connection to another proxy. This must have previously been specified in cache_peer. If all is specified as the ACL_NAME, all requests are forwarded directly to the parent. This can be necessary, for example, if you are using a provider that dictates the use of its proxies or denies its firewall direct Internet access.

31.4.2 Options for Access Controls

Squid provides a detailed system for controlling the access to the proxy. These Access Control Lists (ACL) are lists with rules that are processed sequentially. ACLs must be defined before they can be used. Some default ACLs, such as `all` and `localhost`, already exist. However, the mere definition of an ACL does not mean that it is actually applied. This only happens when there is a corresponding `http_access` rule.

The syntax for the option `acl` is as follows:

```
acl ACL_NAME TYPE DATA
```

The placeholders within this syntax stand for the following:

- The name `ACL_NAME` can be chosen arbitrarily.
- For `TYPE`, select from a variety of different options which can be found in the `ACCESS CONTROLS` section in the `/etc/squid/squid.conf` file.
- The specification for `DATA` depends on the individual ACL type and can also be read from a file. For example, “via” host names, IP addresses, or URLs.

For more information on types of ACL rules, see the Squid documentation at <http://www.squid-cache.org/Versions/v3/3.5/cfgman/acl.html>.

EXAMPLE 31.2: DEFINING ACL RULES

```
acl mysurfers srcdomain .example.com ❶  
acl teachers src 192.168.1.0/255.255.255.0 ❷  
acl students src 192.168.7.0-192.168.9.0/255.255.255.0 ❸  
acl lunch time MTWHF 12:00-15:00 ❹
```

- ❶ This ACL defines `mysurfers` to be all users coming from within `.example.com` (as determined by a reverse lookup for the IP).
- ❷ This ACL defines `teachers` to be the users of computers with IP addresses starting with `192.168.1.`
- ❸ This ACL defines `students` to be the users of the computer with IP addresses starting with `192.168.7.`, `192.168.8.`, or `192.168.9.`
- ❹ This ACL defines `lunch`, as a time on the days Monday, Tuesday, ... Friday between noon and 3 p.m.

http_access allow ACL_NAME

http_access defines who is allowed to use the proxy and who can access what on the Internet. For this, ACLs must be defined. localhost and all have already been defined above for which you can deny or allow access via deny or allow. A list containing any number of http_access entries can be created, processed from top to bottom. Depending on which occurs first, access is allowed or denied to the respective URL. The last entry should always be http_access deny all. In the following example, localhost has free access to everything while all other hosts are denied access completely:

```
http_access allow localhost
http_access deny all
```

In another example using these rules, the group teachers always has access to the Internet. The group students only has access between Monday and Friday during lunch time:

```
http_access deny localhost
http_access allow teachers
http_access allow students lunch time
http_access deny all
```

For readability, within the configuration file /etc/squid/squid.conf, specify all http_access options as a block.

url_rewrite_program PATH

With this option, specify a URL rewriter. For example, this can be squidGuard (/usr/sbin/squidGuard) which allows blocking unwanted URLs. With it, Internet access can be individually controlled for various user groups using proxy authentication and the appropriate ACLs.

For more information on squidGuard, see [Section 31.7, “squidGuard”](#).

auth_param basic program PATH

If users must be authenticated on the proxy, set a corresponding program, such as /usr/sbin/pam_auth. When accessing pam_auth for the first time, the user sees a login window in which they need to specify a user name and a password. In addition, you need an ACL, so only clients with a valid login can use the Internet:

```
acl password proxy_auth REQUIRED

http_access allow password
http_access deny all
```


In the `acl proxy_auth` option, using `REQUIRED` means that all valid user names are accepted. `REQUIRED` can also be replaced with a list of permitted user names.

`ident_lookup_access allow ACL_NAME`

With this option, have an ident request run to find each user's identity for all clients defined by an ACL of the type `src`. Alternatively, use this for all clients, apply the predefined ACL `all` as the `ACL_NAME`.

All clients covered by `ident_lookup_access` must run an ident daemon. On Linux, you can use `pidentd` (package `pidentd`) as the ident daemon. For other operating systems, free software is usually available. To ensure that only clients with a successful ident lookup are permitted, define a corresponding ACL:

```
acl identhsts ident REQUIRED

http_access allow identhsts
http_access deny all
```

In the `acl identhsts ident` option, using `REQUIRED` means that all valid user names are accepted. `REQUIRED` can also be replaced with a list of permitted user names.

Using `ident` can slow down access time, because ident lookups are repeated for each request.

31.5 Configuring a Transparent Proxy

The usual way of working with proxy servers is the following: the Web browser sends requests to a certain port of the proxy server and the proxy always provides these required objects, regardless of whether they are in its cache. However, in some cases the transparent proxy mode of Squid makes sense:

- If, for security reasons, it is recommended that all clients use a proxy to surf the Internet.
- If all clients must use a proxy, regardless of whether they are aware of it.
- If the proxy in a network is moved, but the existing clients need to retain their old configuration.

A transparent proxy intercepts and answers the requests of the Web browser, so the Web browser receives the requested pages without knowing where they are coming from. As the name indicates, the entire process is transparent to the user.

PROCEDURE 31.1: SQUID AS A TRANSPARENT PROXY

1. In `/etc/squid/squid.conf`, on the line of the option `http_port` add the parameter `transparent`:

```
http_port 3128 transparent
```

2. Restart Squid:

```
tux > sudo systemctl restart squid
```

3. Set up SuSEFirewall2 to redirect HTTP traffic to the port given in `http_proxy` (in the example above, that was port 3128). To do so, edit the configuration file `/etc/sysconfig/SuSEfirewall2`.

This example assumes that you are using the following devices:

- Device pointing to the Internet: `FW_DEV_EXT="eth1"`
- Device pointing to the network: `FW_DEV_INT="eth0"`

Define ports and services (see `/etc/services`) on the firewall that are accessed from untrusted (external) networks such as the Internet. In this example, only Web services are offered to the outside:

```
FW_SERVICES_EXT_TCP="www"
```

Define ports or services (see `/etc/services`) on the firewall that are accessed from the secure (internal) network, both via TCP and UDP:

```
FW_SERVICES_INT_TCP="domain www 3128"  
FW_SERVICES_INT_UDP="domain"
```

This allows accessing Web services and Squid (whose default port is `3128`). The service “domain” stands for DNS (domain name service). This service is commonly used. Otherwise, simply remove `domain` from the above entries and set the following option to `no`:

```
FW_SERVICE_DNS="yes"
```

The option `FW_REDIRECT` is very important, as it is used for the actual redirection of HTTP traffic to a specific port. The configuration file explains the syntax in a comment above the option:

```
# Format:
# list of <source network>[,<destination network>,<protocol>[,dport[:lport]]
# Where protocol is either tcp or udp. dport is the original
# destination port and lport the port on the local machine to
# redirect the traffic to
#
# An exclamation mark in front of source or destination network
# means everything EXCEPT the specified network
```

That is:

1. Specify the IP address and the netmask of the internal networks accessing the proxy firewall.
2. Specify the IP address and the netmask to which these clients send their requests. In the case of Web browsers, specify the networks `0/0`, a wild card that means “to everywhere.”
3. Specify the original port to which these requests are sent.
4. Specify the port to which all these requests are redirected. In the example below, only Web services (port `80`) are redirected to the proxy port (port `3128`). If there are more networks or services to add, separate them with a space in the respective entry. Because Squid supports protocols other than HTTP, you can also redirect requests from other ports to the proxy. For example, you can also redirect port 21 (FTP) and port 443 (HTTPS or SSL).

Therefore, for a Squid configuration, you could use:

```
FW_REDIRECT="192.168.0.0/16,0/0,tcp,80,3128"
```

4. In the configuration file `/etc/sysconfig/SuSEfirewall2`, make sure that the entry `START_FW` is set to `"yes"`.
5. Restart SuSEFirewall2:

```
tux > sudo systemctl restart SuSEfirewall2
```

6. To verify that everything is working properly, check the Squid log files in `/var/log/squid/access.log`. To verify that all ports are correctly configured, perform a port scan on the machine from any computer outside your network. Only the Web services (port 80) should be open. To scan the ports with `nmap`, use:

```
nmap -0 IP_ADDRESS
```

31.6 Using the Squid Cache Manager CGI Interface (`cachemgr.cgi`)

The Squid cache manager CGI interface (`cachemgr.cgi`) is a CGI utility for displaying statistics about the memory usage of a running Squid process. It is also a convenient way to manage the cache and view statistics without logging the server.

PROCEDURE 31.2: **SETTING UP** `cachemgr.cgi`

1. Make sure the Apache Web server is running on your system. Configure Apache as described in *Chapter 29, The Apache HTTP Server*. In particular, see *Section 29.5, "Enabling CGI Scripts"*. To check whether Apache is already running, use:

```
tux > sudo systemctl status apache2
```

If `inactive` is shown, you can start Apache with the SUSE Linux Enterprise Server default settings:

```
tux > sudo systemctl start apache2
```

2. Now enable `cachemgr.cgi` in Apache. To do so, create a configuration file for a `ScriptAlias`.

Create the file in the directory `/etc/apache2/conf.d` and name it `cachemgr.conf`. In it, add the following:

```
ScriptAlias /squid/cgi-bin/ /usr/lib64/squid/

<Directory "/usr/lib64/squid/">
Options +ExecCGI
AddHandler cgi-script .cgi
Require host HOST_NAME
```

```
</Directory>
```

Replace HOST_NAME with the host name of the computer you want to access cachemgr.cgi from. This allows only your computer to access cachemgr.cgi. To allow access from anywhere, use Require all granted instead.

3.
 - If Squid and your Apache Web server run on the same computer, there should be no changes that need to be made to /etc/squid/squid.conf. However, verify that /etc/squid/squid.conf contains the following lines:

```
http_access allow manager localhost
http_access deny manager
```

These lines allow you to access the manager interface from your own computer (localhost) but not from elsewhere.

- If Squid and your Apache Web server run on different computers, you need to add extra rules to allow access from the CGI script to Squid. Define an ACL for your server (replace WEB_SERVER_IP with the IP address of your Web server):

```
acl webserver src WEB_SERVER_IP/255.255.255.255
```

Make sure the following rules are in the configuration file. Compared to the default configuration, only the rule in the middle is new. However, the sequence is important.

```
http_access allow manager localhost
http_access allow manager webserver
http_access deny manager
```

4. Optionally, you can configure one or more passwords for cachemgr.cgi. This also allows access to more actions such as closing the cache remotely or viewing more information about the cache. For this, configure the options cache_mgr and cachemgr_passwd with one or more password for the manager and a list of allowed actions.

For example, to explicitly enable viewing the index page, the menu, 60-minute average of counters without authentication, to enable toggling offline mode using the password secretpassword, and to completely disable everything else, use the following configuration:

```
cache_mgr user
```

```
cachemgr_passwd none index menu 60min
cachemgr_passwd secretpassword offline_toggle
cachemgr_passwd disable all
```

cache_mgr defines a user name. cache_mgr defines which actions are allowed using which password.

The keywords none and disable are special: none removes the need for a password, disable disables functionality outright.

The full list of actions can be best seen after logging in to cachemgr.cgi. To find out how the operation needs to be referenced in the configuration file, see the string after &operation= in the URL of the action page. all is a special keyword meaning all actions.

5. Reload Squid and Apache after the configuration file changes:

```
tux > sudo systemctl reload squid
```

6. To view the statistics, go to the cachemgr.cgi page that you set up before. For example, it could be <http://webserver.example.org/squid/cgi-bin/cachemgr.cgi>. Choose the right server, and, if set, specify user name and password. Then click *Continue* and browse through the different statistics.

31.7 squidGuard

This section is not intended to explain an extensive configuration of squidGuard, only to introduce it and give some advice for using it. For more in-depth configuration issues, refer to the squidGuard Web site at <http://www.squidguard.org>.

squidGuard is a free (GPL), flexible, and fast filter, redirector, and access controller plug-in for Squid. It lets you define multiple access rules with different restrictions for different user groups on a Squid cache. squidGuard uses Squid's standard redirector interface. squidGuard can do the following:


- Limit Web access for some users to a list of accepted or well-known Web servers or URLs.
- Block access to some listed or blacklisted Web servers or URLs for some users.
- Block access to URLs matching a list of regular expressions or words for some users.
- Redirect blocked URLs to an “intelligent” CGI-based information page.

- Redirect unregistered users to a registration form.
- Redirect banners to an empty GIF.
- Use different access rules based on time of day, day of the week, date, etc.
- Use different rules for different user groups.

squidGuard and Squid cannot be used to:

- Edit, filter, or censor text inside documents.
- Edit, filter, or censor HTML-embedded scripts such as JavaScript.

PROCEDURE 31.3: SETTING UP SQUIDGUARD

1. Before it can be used, install squidGuard .
2. Provide a minimal configuration file as /etc/squidguard.conf . Find configuration examples in <http://www.squidguard.org/Doc/examples.html>  . Experiment later with more complicated configuration settings.
3. Next, create an “access denied” HTML page or CGI page that Squid can redirect to if the client requests a blacklisted Web site. Using Apache is strongly recommended.
4. Now, configure Squid to use squidGuard. Use the following entry in the /etc/squid/squid.conf file:

```
redirect_program /usr/bin/squidGuard
```

5. Another option called redirect_children configures the number of “redirect” (in this case squidGuard) processes running on the machine. The more processes you set, the more RAM is required. Try low numbers first, for example, 4:

```
redirect_children 4
```

6. Last, have Squid load the new configuration by running systemctl reload squid . Now, test your settings with a browser.

31.8 Cache Report Generation with Calamaris

Calamaris is a Perl script used to generate reports of cache activity in ASCII or HTML format. It works with native Squid access log files. The Calamaris home page is located at <http://cord.de/calamaris-english>⁷. This tool does not belong to the SUSE Linux Enterprise Server default installation scope—to use it, install the `calamaris` package.

Log in as `root`, then enter:

```
cat access1.log [access2.log access3.log] | calamaris options > reportfile
```

When using more than one log file, make sure they are chronologically ordered, with older files listed first. This can be achieved by either listing the files one after the other as in the example above, or by using `access{1..3}.log`.

`calamaris` takes the following options:

- `-a`
output all available reports
- `-w`
output as HTML report
- `-l`
include a message or logo in report header

More information about the various options can be found in the program's manual page with `man calamaris`.


A typical example is:

```
cat access.log.{10..1} access.log | calamaris -a -w \  
> /usr/local/httpd/htdocs/Squid/squidreport.html
```

This puts the report in the directory of the Web server. Apache is required to view the reports.

31.9 For More Information

Visit the home page of Squid at <http://www.squid-cache.org/>⁷. Here, find the “Squid User Guide” and a very extensive collection of FAQs on Squid.

In addition, mailing lists are available for Squid at <http://www.squid-cache.org/Support/mailling-lists.html> .

32 Web Based Enterprise Management Using SFCB

32.1 Introduction and Basic Concept

SUSE® Linux Enterprise Server (SLES) provides a collection of open standards based tools for the unified management of disparate computing systems and environments. Our enterprise solutions implement the standards proposed by the Distributed Management Task Force. The following paragraphs describe their basic components.

Distributed Management Task Force, Inc (DMTF) is the industry organization which leads the development of management standards for enterprise and Internet environments. Their goal is to unify management standards and initiatives, and to enable more integrated, cost effective and interoperable management solutions. DMTF standards provide common system management components for control and communication. Their solutions are independent of platforms and technologies. *Web Based Enterprise Management* and *Common Information Model* are one of their key technologies.

Web Based Enterprise Management (WBEM) is a set of management and Internet standard technologies. WBEM is developed to unify the management of enterprise computing environments. It provides the ability for the industry to deliver a well-integrated collection of management tools using Web technologies. WBEM consists of the following standards:

- A data model: the Common Information Model (CIM) standard
- An encoding specification: CIM-XML Encoding Specification
- A transport mechanism: CIM operations over HTTP

Common Information Model is a conceptual information model that describes system management. It is not bound to a particular implementation and enables the interchange of management information between management systems, networks, services and applications. There are two parts to CIM — the CIM Specification and the CIM Schema.

- The *CIM Specification* describes the language, naming and meta schema. The meta schema is a formal definition of the model. It defines the terms used to express the model and their usage and semantics. The elements of the meta schema are *classes*, *properties*, and *methods*. The meta schema also supports *indications* and *associations* as types of *classes*, and *references* as types of *properties*.
- The *CIM Schema* provides the actual model descriptions. It supplies a set of classes with properties and associations that provide a well understood conceptual framework within which it is possible to organize the available information about the managed environment.

The Common Information Model Object Manager (CIMOM) is a CIM object manager or, more specifically, an application that manages objects according to the CIM standard. CIMOM manages communication between CIMOM providers and a CIM client, where the administrator manages the system.

CIMOM providers are software performing specific tasks within the CIMOM that are requested by client applications. Each provider instruments one or more aspects of the CIMOM's schema. These providers interact directly with the hardware.

Standards Based Linux Instrumentation for Manageability (SBLIM) is a collection of tools designed to support Web-Based Enterprise Management (WBEM). SUSE® Linux Enterprise Server uses the open source CIMOM (or CIM server) from the SBLIM project called *Small Footprint CIM Broker*.

Small Footprint CIM Broker is a CIM server intended for use in resource-limited or embedded environments. It is designed to be modular and lightweight at the same time. Its based on open standards and it supports CMPI providers, CIM-XML encoding, and *Managed Object Format (MOF)*. It is highly configurable and performs stability even if the provider crashes. It is also easily accessible as it supports various transport protocols, such as HTTP, HTTPS, Unix domain sockets, Service Location Protocol (SLP), and Java Database Connectivity (JDBC).

32.2 Setting Up SFCB

To set up the Small Footprint CIM Broker (SFCB) environment, make sure the *Web-Based Enterprise Management* pattern in YaST is selected during SUSE Linux Enterprise Server installation. Alternatively, select it as a component to install on a server that is already running. Make sure the following packages are installed on your system:

cim-schema, Common Information Model (CIM) Schema

Contains the Common Information Model (CIM). CIM is a model for describing overall management information in a network or enterprise environments. CIM consists of a specification and a schema. The specification defines the details for integration with other management models. The schema provides the actual model descriptions.

cmapi-bindings-pywbem

Contains an adapter to write and run CMPI-type CIM providers in Python.

cmapi-pywbem-base

Contains base system CIM providers.

cmapi-pywbem-power-management

Contains power management providers based on DSP1027.

python-pywbem

Contains a Python module for making CIM operation calls through the WBEM protocol to query and update managed objects.

cmapi-provider-register, CIMOM neutral provider registration utility

Contains a utility allowing CMPI provider packages to register with whatever CIMOM happens to be present on the system.

sblim-sfcb, Small Footprint CIM Broker

Contains Small Footprint CIM Broker. It is a CIM server conforming to the CIM Operations over HTTP protocol. It is robust, with low resource consumption and, therefore, specifically suited for embedded and resource constrained environments. SFCB supports providers written against the Common Manageability Programming Interface (CMPI).

sblim-sfcc

Contains Small Footprint CIM Client library runtime libraries.

sblim-wbemcli

Contains WBEM command line interface. It is a stand-alone command line WBEM client especially suited for basic systems management tasks.

smis-providers

Contains providers to instrument the volumes and snapshots on the Linux file system. These are based on SNIA's SMI-S volume management profile and Copy Services profile respectively.

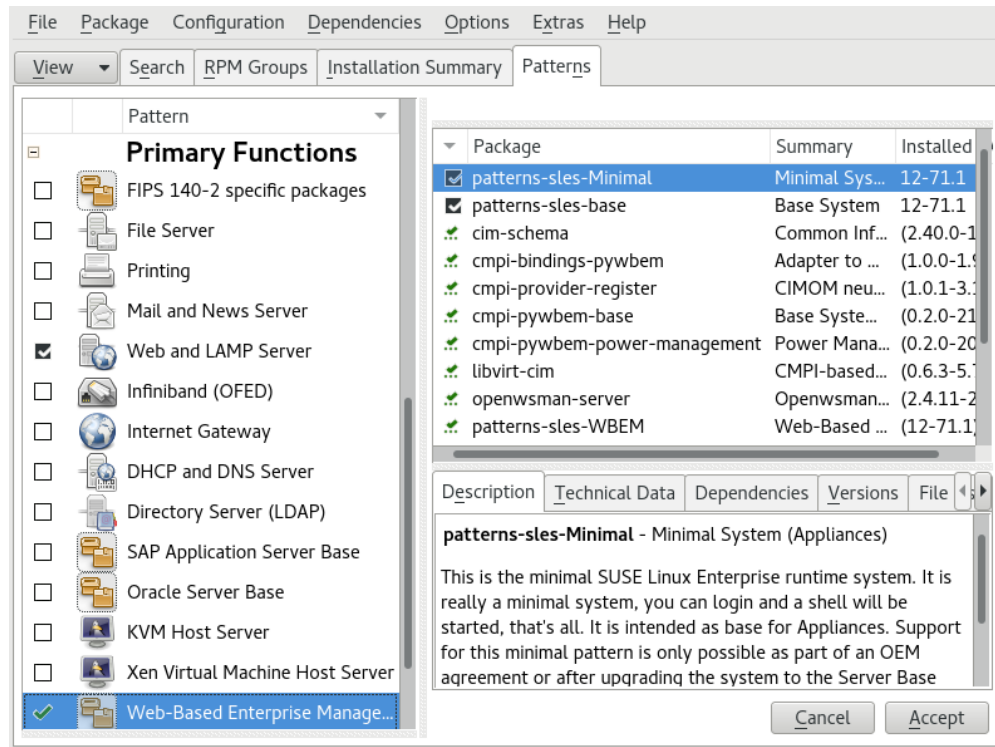


FIGURE 32.1: PACKAGE SELECTION FOR WEB-BASED ENTERPRISE MANAGEMENT PATTERN

32.2.1 Installing Additional Providers

SUSE® Linux Enterprise Server software repository includes additional CIM providers that are not found in the Web-Based Enterprise Management installation pattern. You can easily get their list and installation status by searching the pattern `sblim-cmpi-` in the YaST software installation module. These providers cover various tasks of system management, such as DHCP, NFS, or kernel parameters setting. It is useful to install those providers which you are going to use with SFCB.

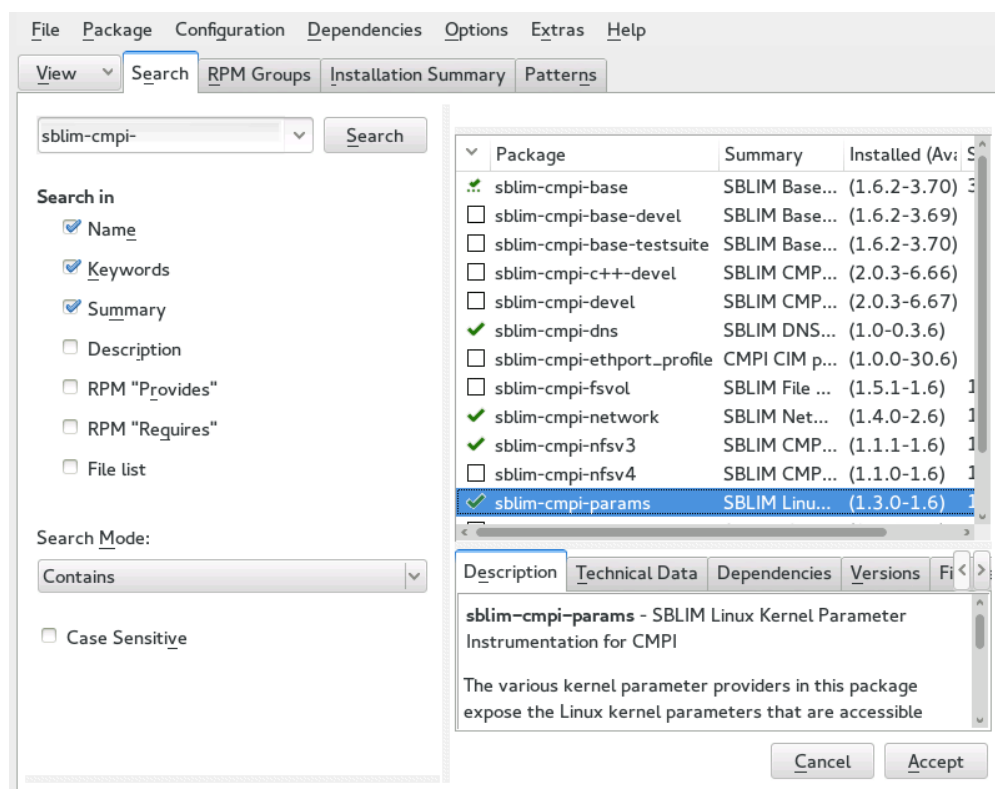


FIGURE 32.2: PACKAGE SELECTION OF ADDITIONAL CIM PROVIDERS

32.2.2 Starting, Stopping and Checking Status for SFCB

CIM server `sfcbd` daemon is installed together with Web-Based Enterprise Management software and is started by default at system start-up. The following table explains how to start, stop and check status for `sfcbd`.

TABLE 32.1: COMMANDS FOR MANAGING SFCBD

Task	Linux Command
Start <code>sfcbd</code>	Enter <code>systemctl start sfcb</code> as <code>root</code> in the command line.
Stop <code>sfcbd</code>	Enter <code>systemctl stop sfcb</code> as <code>root</code> in the command line.
Check <code>sfcbd</code> status	Enter <code>systemctl status sfcb</code> as <code>root</code> in the command line.

32.2.3 Ensuring Secure Access

The default setup of SFCB is relatively secure. However, check that the access to SFCB components is as secure as required for your organization.

32.2.3.1 Certificates

Secure Socket Layers (SSL) transports require a certificate for secure communication to occur. When SFCB is installed, it has a self-signed certificate generated.

You can replace the path to the default certificate with a path to a commercial or self-signed one by changing the `sslCertificateFilePath: path_filename` setting in `/etc/sfcb/sfcb.cfg`. The file must be in PEM format.

The default generated server certificate is in the following location:

/etc/sfcb/server.pem



Note: Paths to SSL Certificates

The default generated certificate files servercert.pem and serverkey.pem are stored under /etc/ssl/servercerts directory. Files /etc/sfcb/client.pem, /etc/sfcb/file.pem and /etc/sfcb/server.pem are symbolic links to these files.

If you want to generate a new certificate, enter the following command as root in the command line:

```
tux > sh /usr/share/sfcb/genSslCert.sh
Generating SSL certificates in .
Generating a 2048 bit RSA private key
.....+++
.+++
writing new private key to '/var/tmp/sfcb.0Bjt69/key.pem'
-----
```

By default, the script generates certificates client.pem, file.pem and server.pem in the current working directory. If you want the script to generate the certificates in /etc/sfcb directory, you need to append it to the command. If these files already exist, a warning message is displayed and the old certificates are not overwritten.

```
tux > sh /usr/share/sfcb/genSslCert.sh /etc/sfcb
```

```
Generating SSL certificates in .  
WARNING: server.pem SSL Certificate file already exists.  
old file will be kept intact.  
WARNING: client.pem SSL Certificate trust store already exists.  
old file will be kept intact.
```

You must remove the old certificates from the file system and run the command again.

If you want to change the way SFCB uses certificates, see [Section 32.2.3.3, “Authentication”](#).

32.2.3.2 Ports

By default, SFCB is configured to accept all communications through the secure port 5989. The following paragraphs explain the communication port setup and recommended configuration.

Port 5989 (secure)

The secure port that SFCB communications use via HTTPS services. This is the default. With this setting, all communications between the CIMOM and client applications are encrypted when sent over the Internet between servers and workstations. Users must authenticate with the client application to reach SFCB server. We recommend that you keep this setting. For the SFCB CIMOM to communicate with the necessary applications, this port must be open on routers and firewalls if they are present between the client application and the nodes being monitored.

Port 5988 (insecure)

The insecure port that SFCB communications use via HTTP services. This setting is disabled by default. With this setting, all communications between the CIMOM and client applications are open for review when sent over the Internet between servers and workstations by anyone, without any authentication. We recommend that you use this setting only when attempting to debug a problem with the CIMOM. As soon as the problem is resolved, disable the non-secure port option back. For the SFCB CIMOM to communicate with the necessary applications that require non-secure access, this port must be open in routers and firewalls between the client application and the nodes being monitored.

If you want to change the default port assignments, see [Section 32.2.3.2, “Ports”](#).

32.2.3.3 Authentication

SFCB supports HTTP basic authentication and authentication based on client certificates (HTTP over SSL connections). Basic HTTP authentication is enabled by specifying `doBasicAuth = true` in the SFCB configuration file (`/etc/sfcb/sfcb.cfg` by default). SUSE® Linux Enterprise Server installation of SFCB supports Pluggable Authentication Modules (PAM) approach; therefore the local root user can authenticate to the SFCB CIMOM with local root user credentials.

If the `sslClientCertificate` configuration property is set to `accept` or `require`, the SFCB HTTP adapter will request a certificate from clients when connecting via HTTP over SSL (HTTPS). If `require` is specified, the client **must** provide a valid certificate (according to the client trust store specified via `sslClientTrustStore`). If the client fails to do so, the connection will be rejected by the CIM server.

The setting `sslClientCertificate = accept` may not be obvious. It is very useful if both basic and client certificate authentication are allowed. If the client can provide a valid certificate, HTTPS connection will be established and the basic authentication procedure will not be executed. If this function cannot verify the certificate, the HTTP basic authentication will take place instead.

32.3 SFCB CIMOM Configuration

SFCB is a lightweight implementation of the CIM server, but it is also highly configurable. Several options can control its behavior. You can control the SFCB server in three ways:

- by setting appropriate environment variables
- by using command line options
- by changing its configuration file

32.3.1 Environment Variables

Several environment variables directly affect the behavior of SFCB. You need to restart the SFCB daemon by `systemctl restart sfcb` for these changes to take effect.

PATH

Specifies the path to the `sfcbd` daemon and utilities.

LD_LIBRARY_PATH

Specifies the path to the sfcbl runtime libraries. Alternatively, you can add this path to the system-wide dynamic loader configuration file /etc/ld.so.conf .

SFCB_PAUSE_PROVIDER

Specifies the provider name. The SFCB server pauses after the provider is loaded for the first time. You can then attach a runtime debugger to the provider's process for debugging purposes.

SFCB_PAUSE_CODEC

Specifies the name of the SFCB codec (currently supports only http). The SFCB server pauses after the codec is loaded for the first time. You can then attach a runtime debugger to the process.

SFCB_TRACE

Specifies the level of debug messages for SFCB. Valid values are 0 (no debug messages), or 1 (key debug messages) to 4 (all debug messages). Default is 1.

SFCB_TRACE_FILE

By default, SFCB outputs its debug messages to standard error output (STDERR). Setting this variable causes the debug messages to be written to a specified file instead.

SBLIM_TRACE

Specifies the level of debug messages for SBLIM providers. Valid values are 0 (no debug messages), or 1 (key debug messages) to 4 (all debug messages).

SBLIM_TRACE_FILE

By default, SBLIM provider outputs its trace messages to STDERR. Setting this variable causes the trace messages to be written to a specified file instead.

32.3.2 Command Line Options

sfcbl, the SFCB daemon, has several command line options that switch particular runtime features on or off. Enter these options when SFCB daemon starts.

-c, --config-file=FILE

When SFCB daemon starts, it reads its configuration from /etc/sfcbl/sfcbl.cfg by default. With this option, you can specify an alternative configuration file.

-d, --daemon

Forces sfcdbd and its child processes to run in the background.

-s, --collect-stats

Turns on runtime statistics collecting. Various sfcdbd runtime statistics will be written to the sfcdbStat file in the current working directory. By default, no statistics are collected.

-l, --syslog-level=LOGLEVEL

Specifies the level of verbosity for the system logging facility. LOGLEVEL can be one of LOG_INFO, LOG_DEBUG, or LOG_ERR, which is the default.

-k, --color-trace=LOGLEVEL

Prints trace output in a different color per process for easier debugging.

-t, --trace-components=NUM

Activates component-level tracing messages, where NUM is an OR-ed bitmask integer that defines which component to trace. After you specify -t ?, it lists all the components and their associated integer bitmask:

```
tux > sfcdbd -t ?
--- Traceable Components:      Int      Hex
---      providerMgr:          1      0x00000001
---      providerDrv:          2      0x00000002
---      cimxmlProc:           4      0x00000004
---      httpDaemon:           8      0x00000008
---      upCalls:              16     0x00000010
---      encCalls:             32     0x00000020
---      ProviderInstMgr:      64     0x00000040
---      providerAssocMgr:    128     0x00000080
---      providers:           256     0x00000100
---      indProvider:         512     0x00000200
---      internalProvider:    1024     0x00000400
---      objectImpl:          2048     0x00000800
---      xmlIn:                4096     0x00010000
---      xmlOut:               8192     0x00020000
---      sockets:             16384     0x00040000
---      memoryMgr:           32768     0x00080000
---      msgQueue:            65536     0x00100000
---      xmlParsing:          131072     0x00200000
---      responseTiming:      262144     0x00400000
---      dbpdaemon:           524288     0x00800000
---      slp:                 1048576     0x01000000
```

A useful value that reveals the internal functions of `sfcdb` but does not generate too many messages, is `-t 2019`.

32.3.3 SFCB Configuration File

SFCB reads its runtime configuration from configuration file `/etc/sfcb/sfcb.cfg` after starting up. This behavior can be overridden using `-c` option at start-up.

The configuration file contains `option : value` pairs, one per line. When making changes to this file, you can use any text editor that saves the file in a format that is native to the environment you are using.

Any setting that has the options commented out with a number sign (`#`) uses the default setting. The following list of options may not be complete. See the content of `/etc/sfcb/sfcb.cfg` and `/usr/share/doc/packages/sblim-sfcb/README` for their complete list.

32.3.3.1 httpPort

Purpose

Specifies the local port value that `sfcdb` should listen to receive HTTP (insecure) requests from CIM clients. Default is `5988`.

Syntax

`httpPort: port_number`

32.3.3.2 enableHttp

Purpose

Specifies whether SFCB should accept HTTP client connections. Default is `false`.

Syntax

enableHttp: option

Option	Description
true	Enables HTTP connections.
false	Disables HTTP connections.

32.3.3.3 httpProcs

Purpose

Specifies the maximum number of simultaneous HTTP client connections before new incoming HTTP requests are blocked. Default is 8 .

Syntax

httpProcs: max_number_of_connections

32.3.3.4 httpUserSFCB, httpUser

Purpose

These options control what user the HTTP server will run under. If httpUserSFCB is true, HTTP will run under the same user as the SFCB main process. If it is false the user name specified for httpUser will be used. This setting is used for both HTTP and HTTPS servers. httpUser *must* be specified if httpUserSFCB is set to false. the default is true.

Syntax

httpUserSFCB: true

32.3.3.5 httpLocalOnly

Purpose

Specifies whether to limit HTTP requests to localhost only. Default is false.

Syntax

httpLocalOnly: false

32.3.3.6 httpsPort

Purpose

Specifies the local port value where sfcdb listens for HTTPS requests from CIM clients. Default is 5989 .

Syntax

httpsPort: port_number

32.3.3.7 enableHttps

Purpose

Specifies if SFCB will accept HTTPS client connections. Default is true .

Syntax

enableHttps: option

Option	Description
true	Enables HTTPS connections.

Option	Description
false	Disables HTTPS connections.

32.3.3.8 httpsProcs

Purpose

Specifies the maximum number of simultaneous HTTPS client connections before new incoming HTTPS requests are blocked. Default is 8 .

Syntax

httpsProcs: max_number_of_connections

32.3.3.9 enableInterOp

Purpose

Specifies if SFCB will provide the *interop* namespace for indication support. Default is true .

Syntax

enableInterOp: option

Option	Description
true	Enables interop namespace.
false	Disables interop namespace.

32.3.3.10 provProcs

Purpose

Specifies the maximum number of simultaneous provider processes. After this point, if a new incoming request requires loading a new provider, then one of the existing providers will first be automatically unloaded. Default is 32 .

Syntax

provProcs: max_number_of_procs

32.3.3.11 doBasicAuth

Purpose

Switches basic authentication on or off based on the client user identifier before it accepts the request. Default value is true which means that basic client authentication is performed.

Syntax

doBasicAuth: option

Option	Description
true	Enables basic authentication.
false	Disables basic authentication.

32.3.3.12 basicAuthLib

Purpose

Specifies the local library name. The SFCB server loads the library to authenticate the client user identifier. Default is sfcBasicPAMAuthentication .

Syntax

provProcs: max_number_of_procs

32.3.3.13 useChunking

Purpose

This option switches the use of HTTP/HTTPS “chunking” on or off. If switched on, the server will return large volumes of response data to the client in smaller “chunks”, rather than to buffer the data and send it back all in one chunk. Default is true .

Syntax

useChunking: option

Option	Description
true	Enables HTTP/HTTPS data chunking.
false	Disables HTTP/HTTPS data chunking.

32.3.3.14 `keepaliveTimeout`

Purpose

Specifies the maximum time in seconds that SFCB HTTP process waits between two requests on one connection before it terminates. Setting it to 0 disables HTTP keep-alive. Default is 0.

Syntax

keepaliveTimeout: secs

32.3.3.15 `keepaliveMaxRequest`

Purpose

Specifies the maximum number of consecutive requests on one connection. Setting it to 0 disables HTTP keep-alive. Default value is 10 .

Syntax

keepaliveMaxRequest: number_of_connections

32.3.3.16 `registrationDir`

Purpose

Specifies the registration directory, which contains the provider registration data, the staging area, and the static repository. Default is /var/lib/sfcb/registration .

Syntax

registrationDir: dir

32.3.3.17 providerDirs

Purpose

Specifies a space-separated list of directories where SFCB is searching for provider libraries.
Default is /usr/lib64 /usr/lib64 /usr/lib64/cmpi.

Syntax

providerDirs: dir

32.3.3.18 providerSampleInterval

Purpose

Specifies the interval in seconds at which the provider manager is checking for idle providers.
Default is 30.

Syntax

providerSampleInterval: secs

32.3.3.19 providerTimeoutInterval

Purpose

Specifies the interval in seconds before an idle provider gets unloaded by the provider manager.
Default is 60.

Syntax

providerTimeoutInterval: secs

32.3.3.20 providerAutoGroup

Purpose

If the provider registration file does not specify any other group, and the option is set to true, all providers in the same shared library are executed in the same process.

Syntax

providerAutoGroup: option

Option	Description
true	Enables grouping of providers.
false	Disables grouping of providers.

32.3.3.21 sslCertificateFilePath

Purpose

Specifies the name of the file that contains the server certificate. The file must be in PEM (Privacy Enhanced Mail, RFC 1421 and RFC 1424) format. This file is only required if enableHttps is set to true. Default is /etc/sfcb/server.pem.

Syntax

sslCertificateFilePath: path

32.3.3.22 `sslKeyFilePath`

Purpose

Specifies the name of the file that contains the private key for the server certificate. The file must be in PEM format and may not be protected by passphrase. This file is only required if `enableHttps` is set to `true`. Default is `/etc/sfcb/file.pem`.

Syntax

`sslKeyFilePath:` *path*

32.3.3.23 `sslClientTrustStore`

Purpose

Specifies the name of the file that contains either the CA or self-signed certificates of the clients. This file must be in PEM format and is only required if `sslClientCertificate` is set to `accept` or `require`. Default is `/etc/sfcb/client.pem`.

Syntax

`sslClientTrustStore:` *path*

32.3.3.24 `sslClientCertificate`

Purpose

Specifies the way SFCB handles client certificate based authentication. If set to `ignore`, it will not request a certificate from the client. If set to `accept` it will request a certificate from the client but will not fail if the client does not present one. If set to `require`, it will refuse the client connection if the client does not present a certificate. Default value is `ignore`.

Syntax

sslClientCertificate: option

Option	Description
ignore	Disables requesting a client certificate.
accept	Disables requesting a client certificate. Will not fail if no certificate is present.
require	Refuses the client connection without a valid certificate.

32.3.3.25 certificateAuthLib

Purpose

Specifies the name of the local library to request for the user authentication based on client certificate. This is only requested if sslClientCertificate is not set to ignore. Default value is sfcCertificateAuthentication.

Syntax

certificateAuthLib: file

32.3.3.26 traceLevel

Purpose

Specifies the trace level for SFCB. You can override it by setting environment variable SFCB_TRACE_LEVEL. Default value is 0.

Syntax

traceLevel: num_level

32.3.3.27 traceMask

Purpose

Specifies the trace mask for SFCB. you can override it by the command line option --trace-components. Default value is 0.

Syntax

traceMask: mask

32.3.3.28 traceFile

Purpose

Specifies the trace file for SFCB. You can override it by setting environment variable SFCB_TRACE_FILE. Default value is stderr (standard error output).

Syntax

traceFile: output

32.4 Advanced SFCB Tasks

This chapter covers more advanced topics related to SFCB usage. To understand them, you need to have basic knowledge of the Linux file system and experience with the Linux command line. This chapter includes the following tasks:

- Installing CMPI providers
- Testing SFCB
- Using wbemcli CIM client

32.4.1 Installing CMPI Providers

To install a CMPI provider, you need to make sure that its shared library is copied into one of the directories specified by providerDirs configuration option, see [Section 32.3.3.17, “providerDirs”](#). The provider must also be properly registered using sfcbstage and sfcbrepos commands. The provider package is usually prepared for SFCB, so that its installation takes care of the proper registration. Most SBLIM providers are prepared for SFCB.

32.4.1.1 Class Repository

Class repository is a place where SFCB stores information about CIM classes. It usually consists of a directory tree with namespace components. Typical CIM namespaces are root/cimv2 or root/interop, which respectively translate to the class repository directory path on the file system

/var/lib/sfcb/registration/repository/root/cimv2

and

/var/lib/sfcb/registration/repository/root/interop

Each namespace directory contains the file classSchemas. The file has a compiled binary representation of all the CIM classes registered under that namespace. It also contains necessary information about their CIM superclasses.

In addition, each namespace directory may contain a file qualifiers which contains all qualifiers for the namespace. When `sfcbd` restarts, the class provider will scan the directory /var/lib/sfcb/registration/repository/ and all its subdirectories to determine the registered namespaces. Then classSchemas files are decoded and the class hierarchy for each namespace is built.

32.4.1.2 Adding New Classes

SFCB cannot make live CIM class manipulations. You need to add, change or remove classes offline and restart SFCB service with **`systemctl restart sfcb`** to register the changes.

To store providers class and registration information, SFCB uses a place called *staging area*. On SUSE® Linux Enterprise Server systems, it is the directory structure under /var/lib/sfcb/stage/.

To add a new provider, you need to:

- Copy the provider class definition files to the ./mofs subdirectory of staging area directory (/var/lib/sfcb/stage/mofs).
- Copy a registration file which contains the name of the class or classes and type of provider, and the name of the executable library file into the ./regs subdirectory.

There are two default “mof” (class definition) files in the staging directory: indication.mof and interop.mof. MOF files under the root stage directory /var/lib/sfcb/stage/mofs will be copied into each namespace after running sfcbrepos command. The interop.mof will only be compiled into the *interop* namespace.

The directory layout may look like the following example:

```
tux > ls /var/lib/sfcb/stage
default.reg  mofs  regs
```

```
tux > ls /var/lib/sfcb/stage/mofs
indication.mof  root
```

```
tux > ls /var/lib/sfcb/stage/mofs/root
cimv2  interop  suse  virt
```

```
tux > ls -l /var/lib/sfcb/stage/mofs/root/cimv2 | less
Linux_ABIPParameter.mof
```

```
Linux_BaseIndication.mof
Linux_Base.mof
Linux_DHCPElementConformsToProfile.mof
Linux_DHCPEntity.mof
[.]
OMC_StorageSettingWithHints.mof
OMC_StorageVolumeDevice.mof
OMC_StorageVolume.mof
OMC_StorageVolumeStorageSynchronized.mof
OMC_SystemStorageCapabilities.mof
```

```
tux > ls -l /var/lib/sfcb/stage/mofs/root/interop
ComputerSystem.mof
ElementConformsToProfile.mof
HostSystem.mof
interop.mof
Linux_DHCPElementConformsToProfile.mof
[.]
OMC_SMIElementSoftwareIdentity.mof
OMC_SMISubProfileRequiresProfile.mof
OMC_SMIVolumeManagementSoftware.mof
ReferencedProfile.mof
RegisteredProfile.mof
```

```
tux > ls -l /var/lib/sfcb/stage/regs
AllocationCapabilities.reg
Linux_ABIPParameter.reg
Linux_BaseIndication.reg
Linux_DHCPGlobal.reg
Linux_DHCPRegisteredProfile.reg
[.]
OMC_Base.sfcb.reg
OMC_CopyServices.sfcb.reg
OMC_PowerManagement.sfcb.reg
OMC_Server.sfcb.reg
RegisteredProfile.reg
```

```
tux > cat /var/lib/sfcb/stage/regs/Linux_DHCPRegisteredProfile.reg
[Linux_DHCPRegisteredProfile]
  provider: Linux_DHCPRegisteredProfileProvider
  location: cmpiLinux_DHCPRegisteredProfile
  type: instance
  namespace: root/interop
#
```

```
[Linux_DHCPElementConformsToProfile]
  provider: Linux_DHCPElementConformsToProfileProvider
  location: cmpiLinux_DHCPElementConformsToProfile
  type: instance association
  namespace: root/cimv2
#
[Linux_DHCPElementConformsToProfile]
  provider: Linux_DHCPElementConformsToProfileProvider
  location: cmpiLinux_DHCPElementConformsToProfile
  type: instance association
  namespace: root/interop
```

SFCB uses a custom provider registration file for each provider.



Note: SBLIM Providers Registration Files

All SBLIM providers on the SBLIM Web site already include a registration file that is used to generate the .reg file for SFCB.

The format of SFCB registration file is:

```
[<class-name>]
  provider: <provide-name>
  location: <library-name>
  type: [instance] [association] [method] [indication]
  group: <group-name>
  unload: never
  namespace: <namespace-for-class> ...
```

where:

<class-name>

The CIM class name (required)

<provider-name>

The CMPI provider name (required)

<location-name>

The name of the provider library (required)

type

The type of the provider (required). This can be any combination of: instance, association, method or indication.

<group-name>

Multiple providers can be grouped together and run under a single process to further minimize runtime resources. All providers registered under the same `<group-name>` will be executed under the same process. By default each provider will be run as a separate process.

unload

Specifies the unload policy for the provider. Currently the only supported option is `never`, which specifies that the provider will not be monitored for idle times and will never be unloaded. By default each provider will be unloaded when its idle times exceed the value specified in the configuration file.

namespace

List of namespaces for which this provider can be executed. This is required, although for most providers this will be `root/cimv2`.

Once all the class definitions and provider registration files are stored in the staging area, you need to rebuild the SFCB class repository with the command `sfcbrepos -f`.

You can add, change or remove classes this way. After rebuilding the class repository, restart SFCB with command `systemctl restart sfcbl`.

Alternatively, the SFCB package contains a utility that will copy provider class mof files and registration files to the correct locations in the staging area.

`sfcbstage -r [provider.reg] [class1.mof] [class2.mof] ...`

After running this command you still need to rebuild the class repository and restart SFCB service.

32.4.2 Testing SFCB

The SFCB package includes two testing scripts: `wbemcat` and `xmltest`.

`wbemcat` sends raw CIM-XML data via HTTP protocol to the specified SFCB host (localhost by default) listening on port 5988. Then it displays the returned results. The following file contains the CIM-XML representation of a standard EnumerateClasses request:

```
<?xml version="1.0" encoding="utf-8"?>
<CIM CIMVERSION="2.0" DTDVERSION="2.0">
  <MESSAGE ID="4711" PROTOCOLVERSION="1.0">
    <SIMPLEREQ>
```

```

<IMETHODCALL NAME="EnumerateClasses">
  <LOCALNAMESPACEPATH>
    <NAMESPACE NAME="root"/>
    <NAMESPACE NAME="cimv2"/>
  </LOCALNAMESPACEPATH>
  <IPARAMVALUE NAME="ClassName">
    <CLASSNAME NAME=""/>
  </IPARAMVALUE>
  <IPARAMVALUE NAME="DeepInheritance">
    <VALUE>TRUE</VALUE>
  </IPARAMVALUE>
  <IPARAMVALUE NAME="LocalOnly">
    <VALUE>FALSE</VALUE>
  </IPARAMVALUE>
  <IPARAMVALUE NAME="IncludeQualifiers">
    <VALUE>FALSE</VALUE>
  </IPARAMVALUE>
  <IPARAMVALUE NAME="IncludeClassOrigin">
    <VALUE>TRUE</VALUE>
  </IPARAMVALUE>
</IMETHODCALL>
</SIMPLEREQ>
</MESSAGE>
</CIM>

```

Sending this request to SFCB CIMOM returns a list of all supported classes for which there is a registered provider. Suppose you save the file as cim_xml_test.xml.

```

tux > wbemcat cim_xml_test.xml | less
HTTP/1.1 200 OK
Content-Type: application/xml; charset="utf-8"
Content-Length: 337565
Cache-Control: no-cache
CIMOperation: MethodResponse

<?xml version="1.0" encoding="utf-8" ?>
<CIM CIMVERSION="2.0" DTDVERSION="2.0">
<MESSAGE ID="4711" PROTOCOLVERSION="1.0">
<SIMPLERSP>
<IMETHODRESPONSE NAME="EnumerateClasses">
[.]
<CLASS NAME="Linux_DHCPPParamsForEntity" SUPERCLASS="CIM_Component">
<PROPERTY.REFERENCE NAME="GroupComponent" REFERENCECLASS="Linux_DHCPEntity">
</PROPERTY.REFERENCE>

```

```

<PROPERTY.REFERENCE NAME="PartComponent" REFERENCECLASS="Linux_DHCPPParams">
</PROPERTY.REFERENCE>
</CLASS>
</IRETURNVALUE>
</IMETHODRESPONSE>
</SIMPLERSP>
</MESSAGE>
</CIM>

```

The classes listed will vary depending on what providers are installed on your system.

The second script **xmltest** is also used to send a raw CIM-XML test file to the SFCB CIMOM. It then compares the returned results against a previously saved “OK” result file. If there does not yet exist a corresponding “OK” file, it will be created for later use:

```

tux > xmltest cim_xml_test.xml
Running test cim_xml_test.xml ... OK
      Saving response as cim_xml_test.OK
root # xmltest cim_xml_test.xml
Running test cim_xml_test.xml ... Passed

```

32.4.3 Command Line CIM Client: wbemcli

In addition to **wbemcat** and **xmltest**, the SBLIM project includes a more advanced command line CIM client **wbemcli**. The client is used to send CIM requests to SFCB server and display returned results. It is independent of CIMOM library and can be used with all WBEM compliant implementations.

For example, if you need to list all the classes implemented by SBLIM providers registered to your SFCB, send the “EnumerateClasses” (ec) request to SFCB:

```

tux > wbemcli -dx ec http://localhost/root/cimv2
To server: <?xml version="1.0" encoding="utf-8" ?>
<CIM CIMVERSION="2.0" DTDVERSION="2.0">
<MESSAGE ID="4711" PROTOCOLVERSION="1.0"><SIMPLEREQ><IMETHODCALL \
  NAME="EnumerateClasses"><LOCALNAMESPACEPATH><NAMESPACE NAME="root"> \
  </NAMESPACE><NAMESPACE NAME="cimv2"></NAMESPACE> \
  </LOCALNAMESPACEPATH>
<IPARAMVALUE NAME="DeepInheritance"><VALUE>TRUE</VALUE> \
  </IPARAMVALUE>
<IPARAMVALUE NAME="LocalOnly"><VALUE>FALSE</VALUE></IPARAMVALUE>
<IPARAMVALUE NAME="IncludeQualifiers"><VALUE>FALSE</VALUE> \
  </IPARAMVALUE>

```

```

<IPARAMVALUE NAME="IncludeClassOrigin"><VALUE>TRUE</VALUE> \
  </IPARAMVALUE>
</IMETHODCALL></SIMPLEREQ>
</MESSAGE></CIM>
From server: Content-Type: application/xml; charset="utf-8"
From server: Content-Length: 337565
From server: Cache-Control: no-cache
From server: CIMOperation: MethodResponse
From server: <?xml version="1.0" encoding="utf-8" ?>
<CIM CIMVERSION="2.0" DTDVERSION="2.0">
<MESSAGE ID="4711" PROTOCOLVERSION="1.0">
<SIMPLERSP>
<IMETHODRESPONSE NAME="EnumerateClasses">
<IRETURNVALUE>
<CLASS NAME="CIM_ResourcePool" SUPERCLASS="CIM_LogicalElement">
<PROPERTY NAME="Generation" TYPE="uint64">
</PROPERTY>
<PROPERTY NAME="ElementName" TYPE="string">
</PROPERTY>
<PROPERTY NAME="Description" TYPE="string">
</PROPERTY>
<PROPERTY NAME="Caption" TYPE="string">
</PROPERTY>
<PROPERTY NAME="InstallDate" TYPE="datetime">
</PROPERTY>
[.]
<CLASS NAME="Linux_ReiserFileSystem" SUPERCLASS="CIM_UnixLocalFileSystem">
<PROPERTY NAME="FSReservedCapacity" TYPE="uint64">
</PROPERTY>
<PROPERTY NAME="TotalInodes" TYPE="uint64">
</PROPERTY>
<PROPERTY NAME="FreeInodes" TYPE="uint64">
</PROPERTY>
<PROPERTY NAME="ResizeIncrement" TYPE="uint64">
<VALUE>0</VALUE>
</PROPERTY>
<PROPERTY NAME="IsFixedSize" TYPE="uint16">
<VALUE>0</VALUE>
</PROPERTY>
[.]

```

The `-dx` option shows you the actual XML sent to SFCB by `wbemcli` and the actual XML received. In the above example, the first of many returned classes was `CIM_ResourcePool` followed by `Linux_ReiserFileSystem`. Similar entries will appear for all of the other registered classes.

If you omit the `-dx` option, `wbemcli` will display only a compact representation of the returned data:

```
tux > wbemcli ec http://localhost/root/cimv2
localhost:5988/root/cimv2:CIM_ResourcePool Generation=,ElementName=, \
  Description=,Caption=,InstallDate=,Name=,OperationalStatus=, \
  StatusDescriptions=,Status=,HealthState=,PrimaryStatus=, \
  DetailedStatus=,OperatingStatus=,CommunicationStatus=,InstanceID=, \
  PoolID=,Primordial=,Capacity=,Reserved=,ResourceType=, \
  OtherResourceType=,ResourceSubType=, \AllocationUnits=
localhost:5988/root/cimv2:Linux_ReiserFileSystem FSReservedCapacity=, \
  TotalInodes=,FreeInodes=,ResizeIncrement=,IsFixedSize=,NumberOfFiles=, \
  OtherPersistenceType=,PersistenceType=,FileSystemType=,ClusterSize=, \
  MaxFileNameLength=,CodeSet=,CasePreserved=,CaseSensitive=, \
  CompressionMethod=,EncryptionMethod=,ReadOnly=,AvailableSpace=, \
  FileSystemSize=,BlockSize=,Root=,Name=,CreationClassName=,CSName=, \
  CSCreationClassName=,Generation=,ElementName=,Description=,Caption=, \
  InstanceID=,InstallDate=,OperationalStatus=,StatusDescriptions=, \
  Status=,HealthState=,PrimaryStatus=,DetailedStatus=,OperatingStatus= \
  ,CommunicationStatus=,EnabledState=,OtherEnabledState=,RequestedState= \
  ,EnabledDefault=,TimeOfLastStateChange=,AvailableRequestedStates=, \
  TransitioningToState=,PercentageSpaceUse=
[...]
```

32.5 For More Information

FOR MORE DETAILS ABOUT WBEM AND SFCB, SEE THE FOLLOWING SOURCES:

<http://www.dmtf.org> 

Distributed Management Task Force Web site

<http://www.dmtf.org/standards/wbem/> 

Web-Based Enterprise Management (WBEM) Web site

<http://www.dmtf.org/standards/cim/> 

Common Information Model (CIM) Web site

<http://sblim.wiki.sourceforge.net/> ↗

Standards Based Linux Instrumentation (SBLIM) Web site

<http://sblim.sourceforge.net/wiki/index.php/Sfcb> ↗

Small Footprint CIM Broker (SFCB) Web site

<http://sblim.sourceforge.net/wiki/index.php/Providers> ↗

SBLIM providers packages

IV Mobile Computers

- 33 Mobile Computing with Linux **514**
- 34 Using NetworkManager **525**
- 35 Power Management **535**

33 Mobile Computing with Linux

Mobile computing is mostly associated with laptops, PDAs and cellular phones (and the data exchange between them). Mobile hardware components, such as external hard disks, flash disks, or digital cameras, can be connected to laptops or desktop systems. A number of software components are involved in mobile computing scenarios and some applications are tailor-made for mobile use.

33.1 Laptops

The hardware of laptops differs from that of a normal desktop system. This is because criteria like exchangeability, space requirements and power consumption must be taken into account. The manufacturers of mobile hardware have developed standard interfaces like PCMCIA (Personal Computer Memory Card International Association), Mini PCI and Mini PCIe that can be used to extend the hardware of laptops. The standards cover memory cards, network interface cards, and external hard disks.

33.1.1 Power Conservation

The inclusion of energy-optimized system components during laptop manufacturing contributes to their suitability for use without access to the electrical power grid. Their contribution to conservation of power is at least as important as that of the operating system. SUSE® Linux Enterprise Server supports various methods that influence the power consumption of a laptop and have varying effects on the operating time under battery power. The following list is in descending order of contribution to power conservation:

- Throttling the CPU speed.
- Switching off the display illumination during pauses.
- Manually adjusting the display illumination.
- Disconnecting unused, hotplug-enabled accessories (USB CD-ROM, external mouse, unused PCMCIA cards, Wi-Fi, etc.).
- Spinning down the hard disk when idling.

Detailed background information about power management in SUSE Linux Enterprise Server is provided in *Chapter 35, Power Management*.

33.1.2 Integration in Changing Operating Environments

Your system needs to adapt to changing operating environments when used for mobile computing. Many services depend on the environment and the underlying clients must be reconfigured. SUSE Linux Enterprise Server handles this task for you.

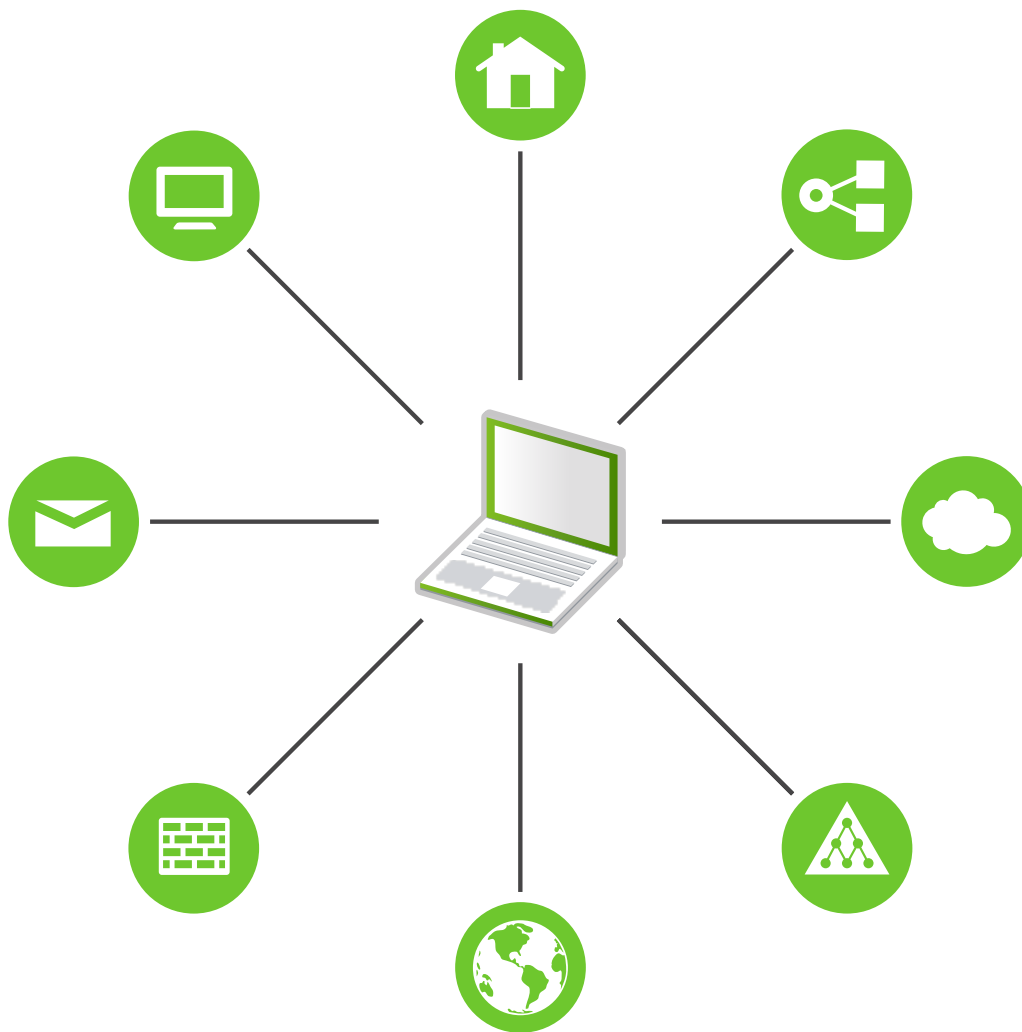


FIGURE 33.1: INTEGRATING A MOBILE COMPUTER IN AN EXISTING ENVIRONMENT

The services affected in the case of a laptop commuting back and forth between a small home network and an office network are:

Network

This includes IP address assignment, name resolution, Internet connectivity and connectivity to other networks.

Printing

A current database of available printers and an available print server must be present, depending on the network.

E-Mail and Proxies

As with printing, the list of the corresponding servers must be current.

X (Graphical Environment)

If your laptop is temporarily connected to a projector or an external monitor, different display configurations must be available.

SUSE Linux Enterprise Server offers several ways of integrating laptops into existing operating environments:

NetworkManager

NetworkManager is especially tailored for mobile networking on laptops. It provides a means to easily and automatically switch between network environments or different types of networks such as mobile broadband (such as GPRS, EDGE, or 3G), wireless LAN, and Ethernet. NetworkManager supports WEP and WPA-PSK encryption in wireless LANs. It also supports dial-up connections. The GNOME desktop includes a front-end for NetworkManager. For more information, see [Section 34.3, "Configuring Network Connections"](#).

TABLE 33.1: USE CASES FOR NETWORKMANAGER

My computer...	Use NetworkManager
is a laptop	Yes
is sometimes attached to different networks	Yes
provides network services (such as DNS or DHCP)	No

My computer...	Use NetworkManager
only uses a static IP address	No

Use the YaST tools to configure networking whenever NetworkManager should not handle network configuration.



Tip: DNS Configuration and Various Types of Network Connections

If you travel frequently with your laptop and change different types of network connections, NetworkManager works fine when all DNS addresses are assigned correctly assigned with DHCP. If some connections use static DNS address(es), add it to the `NETCONFIG_DNS_STATIC_SERVERS` option in `/etc/sysconfig/network/config`.

SLP

The service location protocol (SLP) simplifies the connection of a laptop to an existing network. Without SLP, the administrator of a laptop usually requires detailed knowledge of the services available in a network. SLP broadcasts the availability of a certain type of service to all clients in a local network. Applications that support SLP can process the information dispatched by SLP and be configured automatically. SLP can also be used to install a system, minimizing the effort of searching for a suitable installation source. Find detailed information about SLP in [Chapter 28, SLP](#).

33.1.3 Software Options

There are various task areas in mobile use that are covered by dedicated software: system monitoring (especially the battery charge), data synchronization, and wireless communication with peripherals and the Internet. The following sections cover the most important applications that SUSE Linux Enterprise Server provides for each task.

33.1.3.1 System Monitoring

Two system monitoring tools are provided by SUSE Linux Enterprise Server:

Power Management

Power Management is an application that lets you adjust the energy saving related behavior of the GNOME desktop. You can typically access it via *Computer > Control Center > System > Power Management*.

System Monitor

The *System Monitor* gathers measurable system parameters into one monitoring environment. It presents the output information in three tabs by default. *Processes* gives detailed information about currently running processes, such as CPU load, memory usage, or process ID number and priority. The presentation and filtering of the collected data can be customized—to add a new type of process information, left-click the process table header and choose which column to hide or add to the view. It is also possible to monitor different system parameters in various data pages or collect the data of various machines in parallel over the network. The *Resources* tab shows graphs of CPU, memory and network history and the *File System* tab lists all partitions and their usage.

33.1.3.2 Synchronizing Data

When switching between working on a mobile machine disconnected from the network and working at a networked workstation in an office, it is necessary to keep processed data synchronized across all instances. This could include e-mail folders, directories and individual files that need to be present for work on the road and at the office. The solution in both cases is as follows:

Synchronizing E-Mail

Use an IMAP account for storing your e-mails in the office network. Then access the e-mails from the workstation using any disconnected IMAP-enabled e-mail client, like Mozilla Thunderbird or Evolution as described in *Book “GNOME User Guide”*. The e-mail client must be configured so that the same folder is always accessed for Sent messages. This ensures that all messages are available along with their status information after the synchronization process has completed. Use an SMTP server implemented in the mail client for sending messages instead of the system-wide MTA postfix or sendmail to receive reliable feedback about unsent mail.

Synchronizing Files and Directories

There are several utilities suitable for synchronizing data between a laptop and a workstation. One of the most widely used is a command-line tool called `rsync`. For more information, see its manual page (`man 1 rsync`).

33.1.3.3 Wireless Communication: Wi-Fi

With the largest range of these wireless technologies, Wi-Fi is the only one suitable for the operation of large and sometimes even spatially separate networks. Single machines can connect with each other to form an independent wireless network or access the Internet. Devices called *access points* act as base stations for Wi-Fi-enabled devices and act as intermediaries for access to the Internet. A mobile user can switch among access points depending on location and which access point is offering the best connection. Like in cellular telephony, a large network is available to Wi-Fi users without binding them to a specific location for accessing it.

Wi-Fi cards communicate using the 802.11 standard, prepared by the IEEE organization. Originally, this standard provided for a maximum transmission rate of 2 Mbit/s. Meanwhile, several supplements have been added to increase the data rate. These supplements define details such as the modulation, transmission output, and transmission rates (see *Table 33.2, "Overview of Various Wi-Fi Standards"*). Additionally, many companies implement hardware with proprietary or draft features.

TABLE 33.2: OVERVIEW OF VARIOUS WI-FI STANDARDS

Name (802.11)	Frequency (GHz)	Maximum Transmission Rate (Mbit/s)	Note
a	5	54	Less interference-prone
b	2.4	11	Less common
g	2.4	54	Widespread, backward-compatible with 11b
n	2.4 and/or 5	300	Common

Name (802.11)	Frequency (GHz)	Maximum Transmission Rate (Mbit/s)	Note
ac	5	up to ~865	Expected to be common in 2015
ad	60	up to appr. 7000	Released 2012, currently less common; not supported in SUSE Linux Enterprise Server

802.11 Legacy cards are not supported by SUSE® Linux Enterprise Server. Most cards using 802.11 a/b/g/n are supported. New cards usually comply with the 802.11n standard, but cards using 802.11g are still available.

33.1.3.3.1 Operating Modes

In wireless networking, various techniques and configurations are used to ensure fast, high-quality, and secure connections. Usually your Wi-Fi card operates in *managed mode*. However, different operating types need different setups. Wireless networks can be classified into four network modes:

Managed Mode (Infrastructure Mode), via Access Point (default mode)

Managed networks have a managing element: the access point. In this mode (also called infrastructure or default mode), all connections of the Wi-Fi stations in the network run through the access point, which may also serve as a connection to an Ethernet. To make sure only authorized stations can connect, various authentication mechanisms (WPA, etc) are used. This is also the main mode that consumes the least amount of energy.

Ad-hoc Mode (Peer-to-Peer Network)

Ad-hoc networks do not have an access point. The stations communicate directly with each other, therefore an ad-hoc network is usually slower than a managed network. However, the transmission range and number of participating stations are greatly limited in ad-hoc networks. They also do not support WPA authentication. If you intend to use WPA security, you should not use ad-hoc mode. Be aware, not all cards support ad-hoc mode reliably.

Master Mode

In master mode, your Wi-Fi card is used as the access point, assuming your card supports this mode. Find out the details of your Wi-Fi card at <http://linux-wless.passsys.nl>.

Mesh Mode

Wireless mesh networks are organized in a *mesh topology*. A wireless mesh network's connection is spread among all wireless mesh *nodes*. Each node belonging to this network is connected to other nodes to share the connection, possibly over a large area. (Not supported in SLE12).

33.1.3.3.2 Authentication

Because a wireless network is much easier to intercept and compromise than a wired network, the various standards include authentication and encryption methods.

Old Wi-Fi cards support only WEP (Wired Equivalent Privacy). However, because WEP has proven to be insecure, the Wi-Fi industry has defined an extension called WPA, which is supposed to eliminate the weaknesses of WEP. WPA, sometimes synonymous with WPA2, should be the default authentication method.

Usually the user cannot choose the authentication method. For example, when a card operates in managed mode the authentication is set by the access point. NetworkManager shows the authentication method.

33.1.3.3.3 Encryption

There are various encryption methods to ensure that no unauthorized person can read the data packets that are exchanged in a wireless network or gain access to the network:

WEP (defined in IEEE 802.11)

This standard uses the RC4 encryption algorithm, originally with a key length of 40 bits, later also with 104 bits. Often, the length is declared as 64 bits or 128 bits, depending on whether the 24 bits of the initialization vector are included. However, this standard has some weaknesses. Attacks against the keys generated by this system may be successful. Nevertheless, it is better to use WEP than not to encrypt the network.

Some vendors have implemented the non-standard “Dynamic WEP”. It works exactly as WEP and shares the same weaknesses, except that the key is periodically changed by a key management service.

TKIP (defined in WPA/IEEE 802.11i)

This key management protocol defined in the WPA standard uses the same encryption algorithm as WEP, but eliminates its weakness. Because a new key is generated for every data packet, attacks against these keys are fruitless. TKIP is used together with WPA-PSK.

CCMP (defined in IEEE 802.11i)

CCMP describes the key management. Usually, it is used in connection with WPA-EAP, but it can also be used with WPA-PSK. The encryption takes place according to AES and is stronger than the RC4 encryption of the WEP standard.

33.1.3.4 Wireless Communication: Bluetooth

Bluetooth has the broadest application spectrum of all wireless technologies. It can be used for communication between computers (laptops) and PDAs or cellular phones, as can IrDA. It can also be used to connect various computers within range. Bluetooth is also used to connect wireless system components, like a keyboard or a mouse. The range of this technology is, however, not sufficient to connect remote systems to a network. Wi-Fi is the technology of choice for communicating through physical obstacles like walls.

33.1.3.5 Wireless Communication: IrDA

IrDA is the wireless technology with the shortest range. Both communication parties must be within viewing distance of each other. Obstacles like walls cannot be overcome. One possible application of IrDA is the transmission of a file from a laptop to a cellular phone. The short path from the laptop to the cellular phone is then covered using IrDA. Long-range transmission of the file to the recipient is handled by the mobile network. Another application of IrDA is the wireless transmission of printing jobs in the office.

33.1.4 Data Security

Ideally, you protect data on your laptop against unauthorized access in multiple ways. Possible security measures can be taken in the following areas:

Protection against Theft

Always physically secure your system against theft whenever possible. Various securing tools (like chains) are available in retail stores.

Strong Authentication

Use biometric authentication in addition to standard authentication via login and password. SUSE Linux Enterprise Server supports fingerprint authentication.

Securing Data on the System

Important data should not only be encrypted during transmission, but also on the hard disk. This ensures its safety in case of theft. The creation of an encrypted partition with SUSE Linux Enterprise Server is described in *Book “Security Guide”, Chapter 11 “Encrypting Partitions and Files”*. Another possibility is to create encrypted home directories when adding the user with YaST.



Important: Data Security and Suspend to Disk

Encrypted partitions are not unmounted during a suspend to disk event. Thus, all data on these partitions is available to any party who manages to steal the hardware and issue a resume of the hard disk.

Network Security

Any transfer of data should be secured, no matter how the transfer is done. Find general security issues regarding Linux and networks in *Book “Security Guide”, Chapter 1 “Security and Confidentiality”*.

33.2 Mobile Hardware

SUSE Linux Enterprise Server supports the automatic detection of mobile storage devices over FireWire (IEEE 1394) or USB. The term *mobile storage device* applies to any kind of FireWire or USB hard disk, flash disk, or digital camera. These devices are automatically detected and configured when they are connected with the system over the corresponding interface. The file manager of GNOME offers flexible handling of mobile hardware items. To unmount any of these media safely, use the *Unmount Volume* (GNOME) feature of the file manager. For more details refer to *Book “GNOME User Guide”*.

External Hard Disks (USB and FireWire)

When an external hard disk is correctly recognized by the system, its icon appears in the file manager. Clicking the icon displays the contents of the drive. It is possible to create directories and files here and edit or delete them. To rename a hard disk from the name

it was given by the system, select the corresponding menu item from the menu that opens when the icon is right-clicked. This name change is limited to display in the file manager. The descriptor by which the device is mounted in /media remains unaffected by this.

USB Flash Disks

These devices are handled by the system like external hard disks. It is similarly possible to rename the entries in the file manager.

33.3 Cellular Phones and PDAs

A desktop system or a laptop can communicate with a cellular phone via Bluetooth or IrDA. Some models support both protocols and some only one of the two. The usage areas for the two protocols and the corresponding extended documentation has already been mentioned in *Section 33.1.3.3, "Wireless Communication: Wi-Fi"*. The configuration of these protocols on the cellular phones themselves is described in their manuals.

33.4 For More Information

The central point of reference for all questions regarding mobile devices and Linux is <http://tuxmobil.org/>. Various sections of that Web site deal with the hardware and software aspects of laptops, PDAs, cellular phones and other mobile hardware.

A similar approach to that of <http://tuxmobil.org/> is made by <http://www.linux-on-laptops.com/>. Information about laptops and handhelds can be found here.

SUSE maintains a mailing list in German dedicated to the subject of laptops. See <http://lists.opensuse.org/opensuse-mobile-de/>. On this list, users and developers discuss all aspects of mobile computing with SUSE Linux Enterprise Server. Postings in English are answered, but the majority of the archived information is only available in German. Use <http://lists.opensuse.org/opensuse-mobile/> for English postings.

34 Using NetworkManager

NetworkManager is the ideal solution for laptops and other portable computers. It supports state-of-the-art encryption types and standards for network connections, including connections to 802.1X protected networks. 802.1X is the “IEEE Standard for Local and Metropolitan Area Networks—Port-Based Network Access Control”. With NetworkManager, you need not worry about configuring network interfaces and switching between wired or wireless networks when you are moving. NetworkManager can automatically connect to known wireless networks or manage several network connections in parallel—the fastest connection is then used as default. Furthermore, you can manually switch between available networks and manage your network connection using an applet in the system tray.

Instead of only one connection being active, multiple connections may be active at once. This enables you to unplug your laptop from an Ethernet and remain connected via a wireless connection.

34.1 Use Cases for NetworkManager

NetworkManager provides a sophisticated and intuitive user interface, which enables users to easily switch their network environment. However, NetworkManager is not a suitable solution in the following cases:

- Your computer provides network services for other computers in your network, for example, it is a DHCP or DNS server.
- Your computer is a Xen server or your system is a virtual system inside Xen.

34.2 Enabling or Disabling NetworkManager

On laptop computers, NetworkManager is enabled by default. However, it can be at any time enabled or disabled in the YaST Network Settings module.

1. Run YaST and go to *System > Network Settings*.
2. The *Network Settings* dialog opens. Go to the *Global Options* tab.

3. To configure and manage your network connections with NetworkManager:
 - a. In the *Network Setup Method* field, select *User Controlled with NetworkManager*.
 - b. Click *OK* and close YaST.
 - c. Configure your network connections with NetworkManager as described in [Section 34.3, "Configuring Network Connections"](#).
4. To deactivate NetworkManager and control the network with your own configuration
 - a. In the *Network Setup Method* field, choose *Controlled by wicked*.
 - b. Click *OK*.
 - c. Set up your network card with YaST using automatic configuration via DHCP or a static IP address.
Find a detailed description of the network configuration with YaST in [Section 15.4, "Configuring a Network Connection with YaST"](#).

34.3 Configuring Network Connections

After having enabled NetworkManager in YaST, configure your network connections with the NetworkManager front-end available in GNOME. It shows tabs for all types of network connections, such as wired, wireless, mobile broadband, DSL, and VPN connections.

To open the network configuration dialog in GNOME, open the settings menu via the status menu and click the *Network* entry.



Note: Availability of Options

Depending on your system setup, you may not be allowed to configure connections. In a secured environment, some options may be locked or require root permission. Ask your system administrator for details.

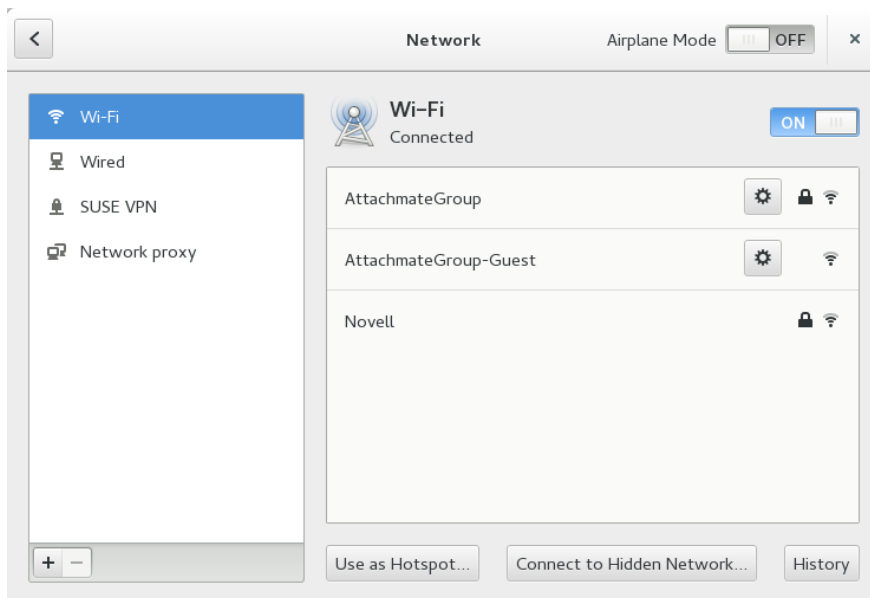


FIGURE 34.1: GNOME NETWORK CONNECTIONS DIALOG

PROCEDURE 34.1: ADDING AND EDITING CONNECTIONS

1. Open the NetworkManager configuration dialog.
2. To add a Connection:
 - a. Click the + icon in the lower left corner.
 - b. Select your preferred connection type and follow the instructions.
 - c. When you are finished click *Add*.
 - d. After having confirmed your changes, the newly configured network connection appears in the list of available networks you get by opening the Status Menu.
3. To edit a connection:
 - a. Select the entry to edit.
 - b. Click the gear icon to open the *Connection Settings* dialog.
 - c. Insert your changes and click *Apply* to save them.
 - d. To Make your connection available as system connection go to the *Identity* tab and set the check box *Make available to other users*. For more information about User and System Connections, see [Section 34.4.1, “User and System Connections”](#).

34.3.1 Managing Wired Network Connections

If your computer is connected to a wired network, use the NetworkManager applet to manage the connection.

1. Open the Status Menu and click *Wired* to change the connection details or to switch it off.
2. To change the settings click *Wired Settings* and then click the gear icon.
3. To switch off all network connections, activate the *Airplane Mode* setting.

34.3.2 Managing Wireless Network Connections

Visible wireless networks are listed in the GNOME NetworkManager applet menu under *Wireless Networks*. The signal strength of each network is also shown in the menu. Encrypted wireless networks are marked with a shield icon.

PROCEDURE 34.2: CONNECTING TO A VISIBLE WIRELESS NETWORK

1. To connect to a visible wireless network, open the Status Menu and click *Wi-Fi*.
2. Click *Turn On* to enable it.
3. Click *Select Network*, select your Wi-Fi Network and click *Connect*.
4. If the network is encrypted, a configuration dialog opens. It shows the type of encryption the network uses and text boxes for entering the login credentials.

PROCEDURE 34.3: CONNECTING TO AN INVISIBLE WIRELESS NETWORK

1. To connect to a network that does not broadcast its service set identifier (SSID or ESSID) and therefore cannot be detected automatically, open the Status Menu and click *Wi-Fi*.
2. Click *Wi-Fi Settings* to open the detailed settings menu.
3. Make sure your Wi-Fi is enabled and click *Connect to Hidden Network*.
4. In the dialog that opens, enter the SSID or ESSID in *Network Name* and set encryption parameters if necessary.

A wireless network that has been chosen explicitly will remain connected as long as possible. If a network cable is plugged in during that time, any connections that have been set to *Stay connected when possible* will be connected, while the wireless connection remains up.

34.3.3 Configuring Your Wi-Fi/Bluetooth Card as an Access Point

If your Wi-Fi/Bluetooth card supports access point mode, you can use NetworkManager for the configuration.

1. Open the Status Menu and click *Wi-Fi*.
2. Click *Wi-Fi Settings* to open the detailed settings menu.
3. Click *Use as Hotspot* and follow the instructions.
4. Use the credentials shown in the resulting dialog to connect to the hotspot from a remote machine.

34.3.4 NetworkManager and VPN

NetworkManager supports several Virtual Private Network (VPN) technologies. For each technology, SUSE Linux Enterprise Server comes with a base package providing the generic support for NetworkManager. In addition to that, you also need to install the respective desktop-specific package for your applet.

OpenVPN

To use this VPN technology, install:

- NetworkManager-openvpn
- NetworkManager-openvpn-gnome

vpnc (Cisco AnyConnect)

To use this VPN technology, install:

- NetworkManager-vpnc
- NetworkManager-vpnc-gnome

PPTP (Point-to-Point Tunneling Protocol)

To use this VPN technology, install:

- NetworkManager-pptp
- NetworkManager-pptp-gnome

The following procedure describes how to set up your computer as an OpenVPN client using NetworkManager. Setting up other types of VPNs works analogously.

Before you begin, make sure that the package `NetworkManager-openvpn-gnome` is installed and all dependencies have been resolved.

PROCEDURE 34.4: SETTING UP OPENVPN WITH NETWORKMANAGER

1. Open the application *Settings* by clicking the status icons at the right end of the panel and clicking the *wrench and screwdriver* icon. In the window *All Settings*, choose *Network*.
2. Click the **+** icon.
3. Select *VPN* and then *OpenVPN*.
4. Choose the *Authentication* type. Depending on the setup of your OpenVPN server, choose *Certificates (TLS)* or *Password with Certificates (TLS)*.
5. Insert the necessary values into the respective text boxes. For our example configuration, these are:

<i>Gateway</i>	The remote endpoint of the VPN server.
<i>User name</i>	The user (only available when you have selected <i>Password with Certificates (TLS)</i>)
<i>Password</i>	The password for the user (only available when you have selected <i>Password with Certificates (TLS)</i>)
<i>User Certificate</i>	<u>/etc/openvpn/client1.crt</u>
<i>CA Certificate</i>	<u>/etc/openvpn/ca.crt</u>
<i>Private Key</i>	<u>/etc/openvpn/client1.key</u>

6. Finish the configuration with *Add*.
7. To enable the connection, in the panel *Network* of the *Settings* application click the switch button. Alternatively, click the status icons at the right end of the panel, click the name of your VPN and then *Connect*.

34.4 NetworkManager and Security

NetworkManager distinguishes two types of wireless connections, trusted and untrusted. A trusted connection is any network that you explicitly selected in the past. All others are untrusted. Trusted connections are identified by the name and MAC address of the access point. Using the MAC address ensures that you cannot use a different access point with the name of your trusted connection.

NetworkManager periodically scans for available wireless networks. If multiple trusted networks are found, the most recently used is automatically selected. NetworkManager waits for your selection in case that all networks are untrusted.

If the encryption setting changes but the name and MAC address remain the same, NetworkManager attempts to connect, but first you are asked to confirm the new encryption settings and provide any updates, such as a new key.

If you switch from using a wireless connection to offline mode, NetworkManager blanks the SSID or ESSID. This ensures that the card is disconnected.

34.4.1 User and System Connections

NetworkManager knows two types of connections: user and system connections. User connections are connections that become available to NetworkManager when the first user logs in. Any required credentials are asked from the user and when the user logs out, the connections are disconnected and removed from NetworkManager. Connections that are defined as system connection can be shared by all users and are made available right after NetworkManager is started—before any users log in. In case of system connections, all credentials must be provided at the time the connection is created. Such system connections can be used to automatically connect to networks that require authorization. For information how to configure user or system connections with NetworkManager, refer to [Section 34.3, “Configuring Network Connections”](#).

34.4.2 Storing Passwords and Credentials

If you do not want to re-enter your credentials each time you want to connect to an encrypted network, you can use the GNOME Keyring Manager to store your credentials encrypted on the disk, secured by a master password.

NetworkManager can also retrieve its certificates for secure connections (for example, encrypted wired, wireless or VPN connections) from the certificate store. For more information, refer to Book “Security Guide”, Chapter 12 “Certificate Store”.

34.5 Frequently Asked Questions

In the following, find some frequently asked questions about configuring special network options with NetworkManager.

34.5.1. How to tie a connection to a specific device?

By default, connections in NetworkManager are device type-specific: they apply to all physical devices with the same type. If more than one physical device per connection type is available (for example, your machine is equipped with two Ethernet cards), you can tie a connection to a certain device.

To do so in GNOME, first look up the MAC address of your device (use the *Connection Information* available from the applet, or use the output of command line tools like `nm-tool` or `wicked show all`). Then start the dialog for configuring network connections and choose the connection you want to modify. On the *Wired* or *Wireless* tab, enter the *MAC Address* of the device and confirm your changes.

34.5.2. How to specify a certain access point in case multiple access points with the same ESSID are detected?

When multiple access points with different wireless bands (a/b/g/n) are available, the access point with the strongest signal is automatically chosen by default. To override this, use the *BSSID* field when configuring wireless connections.

The Basic Service Set Identifier (BSSID) uniquely identifies each Basic Service Set. In an infrastructure Basic Service Set, the BSSID is the MAC address of the wireless access point. In an independent (ad-hoc) Basic Service Set, the BSSID is a locally administered MAC address generated from a 46-bit random number.

Start the dialog for configuring network connections as described in [Section 34.3, “Configuring Network Connections”](#). Choose the wireless connection you want to modify and click *Edit*. On the *Wireless* tab, enter the BSSID.

34.5.3. How to share network connections to other computers?

The primary device (the device which is connected to the Internet) does not need any special configuration. However, you need to configure the device that is connected to the local hub or machine as follows:

1. Start the dialog for configuring network connections as described in [Section 34.3, “Configuring Network Connections”](#). Choose the connection you want to modify and click *Edit*. Switch to the *IPv4 Settings* tab and from the *Method* drop-down box, activate *Shared to other computers*. That will enable IP traffic forwarding and run a DHCP server on the device. Confirm your changes in NetworkManager.
2. As the DHCP server uses port 67, make sure that it is not blocked by the firewall: On the machine sharing the connections, start YaST and select *Security and Users > Firewall*. Switch to the *Allowed Services* category. If *DHCP Server* is not already shown as *Allowed Service*, select *DHCP Server* from *Services to Allow* and click *Add*. Confirm your changes in YaST.

34.5.4. How to provide static DNS information with automatic (DHCP, PPP, VPN) addresses?

In case a DHCP server provides invalid DNS information (and/or routes), you can override it. Start the dialog for configuring network connections as described in [Section 34.3, “Configuring Network Connections”](#). Choose the connection you want to modify and click *Edit*. Switch to the *IPv4 Settings* tab, and from the *Method* drop-down box, activate *Automatic (DHCP) addresses only*. Enter the DNS information in the *DNS Servers* and *Search Domains* fields. To *Ignore automatically obtained routes* click *Routes* and activate the respective check box. Confirm your changes.

34.5.5. How to make NetworkManager connect to password protected networks before a user logs in?

Define a system connection that can be used for such purposes. For more information, refer to [Section 34.4.1, “User and System Connections”](#).

34.6 Troubleshooting

Connection problems can occur. Some common problems related to NetworkManager include the applet not starting or a missing VPN option. Methods for resolving and preventing these problems depend on the tool used.

NetworkManager Desktop Applet Does Not Start

The applet starts automatically if the network is set up for NetworkManager control. If the applet does not start, check if NetworkManager is enabled in YaST as described in [Section 34.2, “Enabling or Disabling NetworkManager”](#). Then make sure that the NetworkManager-gnome package is also installed.

If the desktop applet is installed but is not running for some reason, start it manually. If the desktop applet is installed but is not running for some reason, start it manually with the command `nm-applet`.

NetworkManager Applet Does Not Include the VPN Option

Support for NetworkManager, applets, and VPN for NetworkManager is distributed in separate packages. If your NetworkManager applet does not include the VPN option, check if the packages with NetworkManager support for your VPN technology are installed. For more information, see [Section 34.3.4, “NetworkManager and VPN”](#).

No Network Connection Available

If you have configured your network connection correctly and all other components for the network connection (router, etc.) are also up and running, it sometimes helps to restart the network interfaces on your computer. To do so, log in to a command line as `root` and run `systemctl restart wickeds`.

34.7 For More Information

More information about NetworkManager can be found on the following Web sites and directories:

NetworkManager Project Page


<http://projects.gnome.org/NetworkManager/> 

Package Documentation

Also check out the information in the following directories for the latest information about NetworkManager and the GNOME applet:

- [/usr/share/doc/packages/NetworkManager/](#),
- [/usr/share/doc/packages/NetworkManager-gnome/](#).

35 Power Management

System z The features and hardware described in this chapter do not exist on IBM z Systems, making this chapter irrelevant for these platforms. 

Power management is especially important on laptop computers, but is also useful on other systems. ACPI (Advanced Configuration and Power Interface) is available on all modern computers (laptops, desktops, and servers). Power management technologies require suitable hardware and BIOS routines. Most laptops and many modern desktops and servers meet these requirements. It is also possible to control CPU frequency scaling to save power or decrease noise.

35.1 Power Saving Functions

Power saving functions are not only significant for the mobile use of laptops, but also for desktop systems. The main functions and their use in ACPI are:

Standby

not supported.

Suspend (to memory)

This mode writes the entire system state to the RAM. Subsequently, the entire system except the RAM is put to sleep. In this state, the computer consumes very little power. The advantage of this state is the possibility of resuming work at the same point within a few seconds without having to boot and restart applications. This function corresponds to the ACPI state S3.

Hibernation (suspend to disk)

In this operating mode, the entire system state is written to the hard disk and the system is powered off. There must be a swap partition at least as big as the RAM to write all the active data. Reactivation from this state takes about 30 to 90 seconds. The state prior to the suspend is restored. Some manufacturers offer useful hybrid variants of this mode, such as RediSafe in IBM Thinkpads. The corresponding ACPI state is S4. In Linux, suspend to disk is performed by kernel routines that are independent from ACPI.



Note: Changed UUID for Swap Partitions when Formatting via **mkswap**

Do not reformat existing swap partitions with **mkswap** if possible. Reformatting with **mkswap** will change the UUID value of the swap partition. Either reformat via YaST (will update `/etc/fstab`) or adjust `/etc/fstab` manually.

Battery Monitor

ACPI checks the battery charge status and provides information about it. Additionally, it coordinates actions to perform when a critical charge status is reached.

Automatic Power-Off

Following a shutdown, the computer is powered off. This is especially important when an automatic shutdown is performed shortly before the battery is empty.

Processor Speed Control

In connection with the CPU, energy can be saved in three different ways: frequency and voltage scaling (also known as PowerNow! or Speedstep), throttling and putting the processor to sleep (C-states). Depending on the operating mode of the computer, these methods can also be combined.

35.2 Advanced Configuration and Power Interface (ACPI)

ACPI was designed to enable the operating system to set up and control the individual hardware components. ACPI supersedes both Power Management Plug and Play (PnP) and Advanced Power Management (APM). It delivers information about the battery, AC adapter, temperature, fan and system events, like “close lid” or “battery low.”

The BIOS provides tables containing information about the individual components and hardware access methods. The operating system uses this information for tasks like assigning interrupts or activating and deactivating components. Because the operating system executes commands stored in the BIOS, the functionality depends on the BIOS implementation. The tables ACPI can detect and load are reported in journald. See [Chapter 14, journalctl: Query the systemd Journal](#) for more information on viewing the journal log messages. See [Section 35.2.2, “Troubleshooting”](#) for more information about troubleshooting ACPI problems.

35.2.1 Controlling the CPU Performance

The CPU can save energy in three ways:

- Frequency and Voltage Scaling
- Throttling the Clock Frequency (T-states)
- Putting the Processor to Sleep (C-states)

Depending on the operating mode of the computer, these methods can be combined. Saving energy also means that the system heats up less and the fans are activated less frequently.

Frequency scaling and throttling are only relevant if the processor is busy, because the most economic C-state is applied anyway when the processor is idle. If the CPU is busy, frequency scaling is the recommended power saving method. Often the processor only works with a partial load. In this case, it can be run with a lower frequency. Usually, dynamic frequency scaling controlled by the kernel on-demand governor is the best approach.

Throttling should be used as the last resort, for example, to extend the battery operation time despite a high system load. However, some systems do not run smoothly when they are throttled too much. Moreover, CPU throttling does not make sense if the CPU has little to do.

For in-depth information, refer to *Book “System Analysis and Tuning Guide”, Chapter 11 “Power Management”*.

35.2.2 Troubleshooting

There are two different types of problems. On one hand, the ACPI code of the kernel may contain bugs that were not detected in time. In this case, a solution will be made available for download. More often, the problems are caused by the BIOS. Sometimes, deviations from the ACPI specification are purposely integrated in the BIOS to circumvent errors in the ACPI implementation of other widespread operating systems. Hardware components that have serious errors in the ACPI implementation are recorded in a blacklist that prevents the Linux kernel from using ACPI for these components.

The first thing to do when problems are encountered is to update the BIOS. If the computer does not boot, one of the following boot parameters may be helpful:

pci=noacpi

Do not use ACPI for configuring the PCI devices.

acpi=ht

Only perform a simple resource configuration. Do not use ACPI for other purposes.

acpi=off

Disable ACPI.



Warning: Problems Booting without ACPI

Some newer machines (especially SMP systems and AMD64 systems) need ACPI for configuring the hardware correctly. On these machines, disabling ACPI can cause problems.

Sometimes, the machine is confused by hardware that is attached over USB or FireWire. If a machine refuses to boot, unplug all unneeded hardware and try again.

Monitor the boot messages of the system with the command `dmesg -T | grep -2i acpi` (or all messages, because the problem may not be caused by ACPI) after booting. If an error occurs while parsing an ACPI table, the most important table—the DSDT (*Differentiated System Description Table*)—can be replaced with an improved version. In this case, the faulty DSDT of the BIOS is ignored. The procedure is described in [Section 35.4, “Troubleshooting”](#).

In the kernel configuration, there is a switch for activating ACPI debug messages. If a kernel with ACPI debugging is compiled and installed, detailed information is issued.

If you experience BIOS or hardware problems, it is always advisable to contact the manufacturers. Especially if they do not always provide assistance for Linux, they should be confronted with the problems. Manufacturers will only take the issue seriously if they realize that an adequate number of their customers use Linux.

35.2.2.1 For More Information

- <http://tldp.org/HOWTO/ACPI-HOWTO/> (detailed ACPI HOWTO, contains DSDT patches)
- <http://www.acpi.info> (Advanced Configuration & Power Interface Specification)
- <http://acpi.sourceforge.net/dsdt/index.php> (DSDT patches by Bruno Ducrot)

35.3 Rest for the Hard Disk

In Linux, the hard disk can be put to sleep entirely if it is not needed or it can be run in a more economic or quieter mode. On modern laptops, you do not need to switch off the hard disks manually, because they automatically enter an economic operating mode whenever they are not needed. However, if you want to maximize power savings, test some of the following methods, using the **hdparm** command.

It can be used to modify various hard disk settings. The option **-y** instantly switches the hard disk to the standby mode. **-Y** puts it to sleep. **hdparm -S x** causes the hard disk to be spun down after a certain period of inactivity. Replace **x** as follows: **0** disables this mechanism, causing the hard disk to run continuously. Values from **1** to **240** are multiplied by 5 seconds. Values from **241** to **251** correspond to 1 to 11 times 30 minutes.

Internal power saving options of the hard disk can be controlled with the option **-B**. Select a value from **0** to **255** for maximum saving to maximum throughput. The result depends on the hard disk used and is difficult to assess. To make a hard disk quieter, use the option **-M**. Select a value from **128** to **254** for quiet to fast.

Often, it is not so easy to put the hard disk to sleep. In Linux, numerous processes write to the hard disk, waking it up repeatedly. Therefore, it is important to understand how Linux handles data that needs to be written to the hard disk. First, all data is buffered in the RAM. This buffer is monitored by the **pdflush** daemon. When the data reaches a certain age limit or when the buffer is filled to a certain degree, the buffer content is flushed to the hard disk. The buffer size is dynamic and depends on the size of the memory and the system load. By default, **pdflush** is set to short intervals to achieve maximum data integrity. It checks the buffer every 5 seconds and writes the data to the hard disk. The following variables are interesting:

/proc/sys/vm/dirty_writeback_centisecs

Contains the delay until a **pdflush** thread wakes up (in hundredths of a second).

/proc/sys/vm/dirty_expire_centisecs

Defines after which timeframe a dirty page should be written out latest. Default is 3000, which means 30 seconds.

/proc/sys/vm/dirty_background_ratio

Maximum percentage of dirty pages until pdflush begins to write them. Default is 5 %.

/proc/sys/vm/dirty_ratio

When the dirty page exceeds this percentage of the total memory, processes are forced to write dirty buffers during their time slice instead of continuing to write.



Warning: Impairment of the Data Integrity

Changes to the pdflush daemon settings endanger the data integrity.

Apart from these processes, journaling file systems, like Btrfs, Ext3, Ext4 and others write their metadata independently from pdflush, which also prevents the hard disk from spinning down.

Another important factor is the way active programs behave. For example, good editors regularly write hidden backups of the currently modified file to the hard disk, causing the disk to wake up. Features like this can be disabled at the expense of data integrity.

In this connection, the mail daemon postfix uses the variable POSTFIX_LAPTOP. If this variable is set to yes, postfix accesses the hard disk far less frequently.

35.4 Troubleshooting

All error messages and alerts are logged in the system journal that can be queried with the command journalctl (see *Chapter 14, journalctl: Query the systemd Journal* for more information). The following sections cover the most common problems.

35.4.1 CPU Frequency Does Not Work

Refer to the kernel sources to see if your processor is supported. You may need a special kernel module or module option to activate CPU frequency control. If the kernel-source package is installed, this information is available in /usr/src/linux/Documentation/cpu-freq/.

35.5 For More Information

- http://en.opensuse.org/SDB:Suspend_to_RAM —How to get Suspend to RAM working
- <http://old-en.opensuse.org/Pm-utils> —How to modify the general suspend framework

V Troubleshooting

- 36 Help and Documentation 543
- 37 Gathering System Information for Support 549
- 38 Common Problems and Their Solutions 574

36 Help and Documentation

SUSE® Linux Enterprise Server comes with various sources of information and documentation, many of which are already integrated into your installed system.

Documentation in /usr/share/doc

This traditional help directory holds various documentation files and release notes for your system. It contains also information of installed packages in the subdirectory packages. Find more detailed information in *Section 36.1, “Documentation Directory”*.

Man Pages and Info Pages for Shell Commands

When working with the shell, you do not need to know the options of the commands by heart. Traditionally, the shell provides integrated help by means of man pages and info pages. Read more in *Section 36.2, “Man Pages”* and *Section 36.3, “Info Pages”*.

Desktop Help Center

The help center of the GNOME desktop (Help) provides central access to the most important documentation resources on your system in searchable form. These resources include online help for installed applications, man pages, info pages, and the SUSE manuals delivered with your product.

Separate Help Packages for Some Applications

When installing new software with YaST, the software documentation is usually installed automatically and appears in the help center of your desktop. However, some applications, such as GIMP, may have different online help packages that can be installed separately with YaST and do not integrate into the help centers.

36.1 Documentation Directory

The traditional directory to find documentation on your installed Linux system is /usr/share/doc. Usually, the directory contains information about the packages installed on your system, plus release notes, manuals, and more.




Note: Contents Depends on Installed Packages

In the Linux world, many manuals and other kinds of documentation are available in the form of packages, like software. How much and which information you find in /usr/share/docs also depends on the (documentation) packages installed. If you cannot find the subdirectories mentioned here, check if the respective packages are installed on your system and add them with YaST, if needed.

36.1.1 SUSE Manuals

We provide HTML and PDF versions of our books in different languages. In the manual subdirectory, find HTML versions of most of the SUSE manuals available for your product. For an overview of all documentation available for your product refer to the preface of the manuals.

If more than one language is installed, /usr/share/doc/manual may contain different language versions of the manuals. The HTML versions of the SUSE manuals are also available in the help center of both desktops. For information on where to find the PDF and HTML versions of the books on your installation media, refer to the SUSE Linux Enterprise Server Release Notes. They are available on your installed system under /usr/share/doc/release-notes/ or online at your product-specific Web page at <http://www.suse.com/doc/> .

36.1.2 Package Documentation

Under packages, find the documentation that is included in the software packages installed on your system. For every package, a subdirectory /usr/share/doc/packages/packageName is created. It often contains README files for the package and sometimes examples, configuration files, or additional scripts. The following list introduces typical files to be found under /usr/share/doc/packages. None of these entries are mandatory and many packages might only include a few of them.

AUTHORS

List of the main developers.

BUGS

Known bugs or malfunctions. Might also contain a link to a Bugzilla Web page where you can search all bugs.

CHANGES ,

ChangeLog

Summary of changes from version to version. Usually interesting for developers, because it is very detailed.

COPYING ,

LICENSE

Licensing information.

FAQ

Question and answers collected from mailing lists or newsgroups.

INSTALL

How to install this package on your system. As the package is already installed by the time you get to read this file, you can safely ignore the contents of this file.

README , README.*

General information on the software. For example, for what purpose and how to use it.

TODO

Things that are not implemented yet, but probably will be in the future.

MANIFEST

List of files with a brief summary.

NEWS

Description of what is new in this version.

36.2 Man Pages

Man pages are an essential part of any Linux system. They explain the usage of a command and all available options and parameters. Man pages can be accessed with man followed by the name of the command, for example, man ls.

Man pages are displayed directly in the shell. To navigate them, move up and down with Page ↑ and Page ↓. Move between the beginning and the end of a document with Home and End. End this viewing mode by pressing Q. Learn more about the man command itself with man man. Man pages are sorted in categories as shown in *Table 36.1, “Man Pages—Categories and Descriptions”* (taken from the man page for man itself).

TABLE 36.1: MAN PAGES—CATEGORIES AND DESCRIPTIONS

Number	Description
1	Executable programs or shell commands
2	System calls (functions provided by the Kernel)
3	Library calls (functions within program libraries)
4	Special files (usually found in <code>/dev</code>)
5	File formats and conventions (<code>/etc/fstab</code>)
6	Games
7	Miscellaneous (including macro packages and conventions), for example, <code>man(7)</code> , <code>groff(7)</code>
8	System administration commands (usually only for <code>root</code>)
9	Kernel routines (nonstandard)

Each man page consists of several parts labeled *NAME* , *SYNOPSIS* , *DESCRIPTION* , *SEE ALSO* , *LICENSING* , and *AUTHOR* . There may be additional sections available depending on the type of command.

36.3 Info Pages

Info pages are another important source of information on your system. Usually, they are more detailed than man pages. They consist of more than command line options and contain sometimes whole tutorials or reference documentation. To view the info page for a certain command, enter `info` followed by the name of the command, for example, `info ls` . You can browse an info page with a viewer directly in the shell and display the different sections, called “nodes”.

Use `Space` to move forward and `<-` to move backward. Within a node, you can also browse with `Page ↑` and `Page ↓` but only `Space` and `<-` will take you also to the previous or subsequent node. Press `Q` to end the viewing mode. Not every command comes with an info page and vice versa.

36.4 Online Resources

In addition to the online versions of the SUSE manuals installed under `/usr/share/doc`, you can also access the product-specific manuals and documentation on the Web. For an overview of all documentation available for SUSE Linux Enterprise Server check out your product-specific documentation Web page at <http://www.suse.com/doc/>.

If you are searching for additional product-related information, you can also refer to the following Web sites:

SUSE Technical Support

The SUSE Technical Support can be found at <http://www.suse.com/support/> if you have questions or need solutions for technical problems.

SUSE Forums

There are several forums where you can dive in on discussions about SUSE products. See <http://forums.suse.com/> for a list.

SUSE Conversations

An online community, which offers articles, tips, Q and A, and free tools to download: <http://www.suse.com/communities/conversations/>

GNOME Documentation

Documentation for GNOME users, administrators and developers is available at <http://library.gnome.org/>.

The Linux Documentation Project

The Linux Documentation Project (TLDP) is run by a team of volunteers who write Linux-related documentation (see <http://www.tldp.org>). It is probably the most comprehensive documentation resource for Linux. The set of documents contains tutorials for beginners, but is mainly focused on experienced users and professional system administrators. TLDP publishes HOWTOs, FAQs, and guides (handbooks) under a free license. Parts of the documentation from TLDP are also available on SUSE Linux Enterprise Server.

You can also try general-purpose search engines. For example, use the search terms Linux CD-RW help or OpenOffice file conversion problem if you have trouble with burning CDs or LibreOffice file conversion.

37 Gathering System Information for Support

For a quick overview of all relevant system information of a machine, SUSE Linux Enterprise Server offers the `hostinfo` package. It also helps system administrators to check for tainted Kernels (that are not supported) or any third-party packages installed on a machine.

In case of problems, a detailed system report may be created with either the `supportconfig` command line tool or the YaST *Support* module. Both will collect information about the system such as: current Kernel version, hardware, installed packages, partition setup, and much more. The result is a TAR archive of files. After opening a Service Request (SR), you can upload the TAR archive to Global Technical Support. It will help to locate the issue you reported and to assist you in solving the problem.

Additionally, you can analyze the `supportconfig` output for known issues to help resolve problems faster. For this purpose, SUSE Linux Enterprise Server provides both an appliance and a command line tool for `Supportconfig Analysis` (SCA).

37.1 Displaying Current System Information

For a quick and easy overview of all relevant system information when logging in to a server, use the package `hostinfo`. After it has been installed on a machine, the console displays the following information to any `root` user that logs in to this machine:

EXAMPLE 37.1: OUTPUT OF `hostinfo` WHEN LOGGING IN AS `root`

```
Hostname:                earth
Current As Of:           Wed 12 Mar 2014 03:57:05 PM CET
Distribution:            SUSE Linux Enterprise Server 12
-Service Pack:           0
Architecture:            x86_64
Kernel Version:          3.12.12-3-default
-Installed:              Mon 10 Mar 2014 03:15:05 PM CET
-Status:                 Not Tainted
Last Updated Package:    Wed 12 Mar 2014 03:56:43 PM CET
-Patches Needed:         0
-Security:               0
-3rd Party Packages:     0
IPv4 Address:            ens3 192.168.1.1
Total/Free/+Cache Memory: 983/95/383 MB (38% Free)
Hard Disk:               /dev/sda 10 GB
```

In case the output shows a tainted Kernel status, see *Section 37.6, “Support of Kernel Modules”* for more details.

37.2 Collecting System Information with Supportconfig

To create a TAR archive with detailed system information that you can hand over to Global Technical Support, use either the **supportconfig** command line tool directly or the YaST *Support* module. The command line tool is provided by the package supportutils which is installed by default. The YaST *Support* module is also based on the command line tool.

37.2.1 Creating a Service Request Number

Supportconfig archives can be generated at any time. However, for handing over the supportconfig data to Global Technical Support, you need to generate a service request number first. You will need it to upload the archive to support.

To create a service request, go to <http://www.novell.com/center/eservice> and follow the instructions on the screen. Write down your 11-digit service request number.



Note: Privacy Statement

SUSE and Novell treat system reports as confidential data. For details about our privacy commitment, see <http://www.novell.com/company/legal/privacy/>.

37.2.2 Upload Targets

After having created a service request number, you can upload your supportconfig archives to Global Technical Support as described in *Procedure 37.1, “Submitting Information to Support with YaST”* or *Procedure 37.2, “Submitting Information to Support from Command Line”*. Use one of the following upload targets:

- US customers: <ftp://ftp.novell.com/incoming>
- EMEA, Europe, the Middle East, and Africa: <ftp://support-ftp.suse.com/in>

Alternatively, you can manually attach the TAR archive to your service request using the service request URL: <http://www.novell.com/center/eservice> .

37.2.3 Creating a Supportconfig Archive with YaST

To use YaST to gather your system information, proceed as follows:

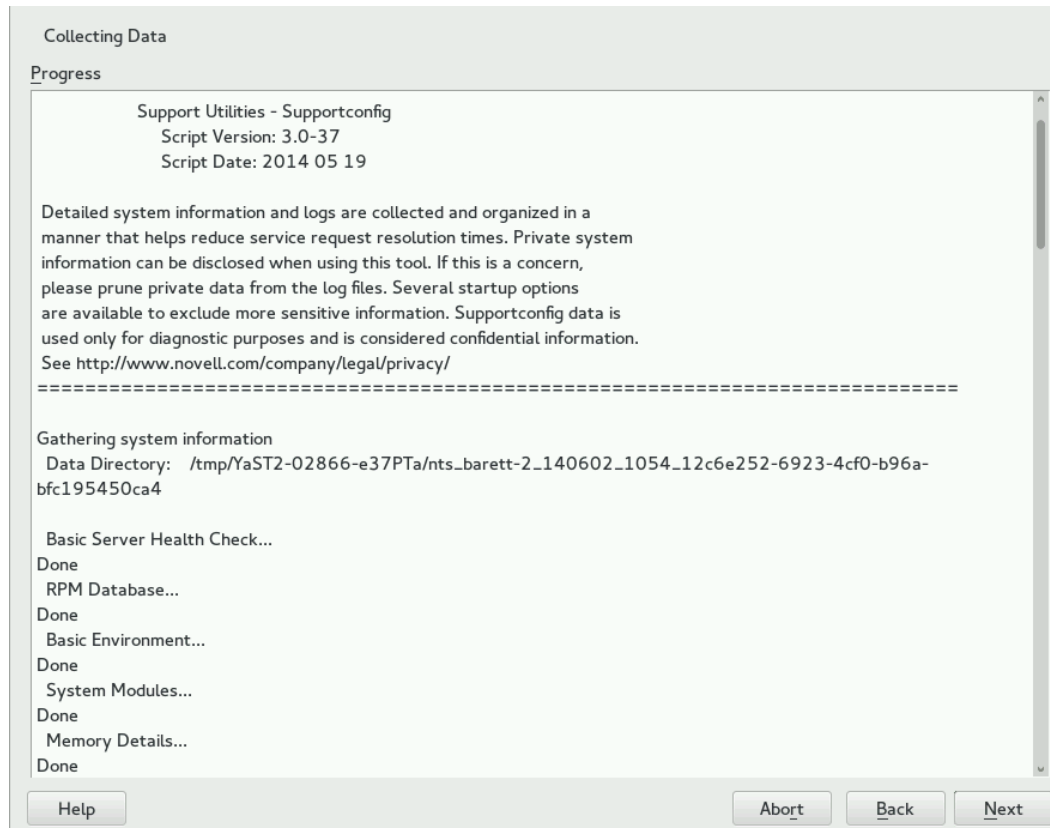
1. Start YaST and open the *Support* module.



2. Click *Create report tarball*.
3. In the next window, select one of the supportconfig options from the radio button list. Use *Custom (Expert) Settings* is preselected by default. If you want to test the report function first, use *Only gather a minimum amount of info*. For some background information on the other options, refer to the [`supportconfig` man page](#). Proceed with *Next*.
4. Enter your contact information. It will be written to a file called `basic-environment.txt` and included in the archive to be created.
5. If you want to submit the archive to Global Technical Support at the end of the information collection process, *Upload Information* is required. YaST automatically proposes an upload server. If you want to modify it, refer to [Section 37.2.2, "Upload Targets"](#) for details of which upload servers are available.

If you want to submit the archive later on, you can leave the *Upload Information* empty for now.

6. Proceed with *Next*.
7. The information gathering begins.



After the process is finished, continue with *Next*.

8. Review the data collection: Select the *File Name* of a log file to view its contents in YaST. To remove any files you want excluded from the TAR archive before submitting it to support, use *Remove from Data*. Continue with *Next*.
9. Save the TAR archive. If you started the YaST module as root user, by default YaST proposes to save the archive to /var/log (otherwise, to your home directory). The file name format is nts_HOST_DATE_TIME.tbz.
10. If you want to upload the archive to support directly, make sure *Upload log files tarball to URL* is activated. The *Upload Target* shown here is the one that YaST proposes in *Step 5*. If you want to modify the upload target, find detailed information of which upload servers are available in *Section 37.2.2, "Upload Targets"*.

11. If you want to skip the upload, deactivate *Upload log files tarball to URL*.
12. Confirm your changes to close the YaST module.

37.2.4 Creating a Supportconfig Archive from Command Line

The following procedure shows how to create a supportconfig archive, but without submitting it to support directly. For uploading it, you need to run the command with certain options as described in *Procedure 37.2, "Submitting Information to Support from Command Line"*.

1. Open a shell and become root.
2. Run **supportconfig** without any options. This gathers the default system information.
3. Wait for the tool to complete the operation.
4. The default archive location is /var/log, with the file name format being nts_HOST_DATE_TIME.tbz

37.2.5 Common Supportconfig Options

The **supportconfig** utility is usually called without any options. Display a list of all options with **supportconfig** -h or refer to the man page. The following list gives a brief overview of some common use cases:

Reducing the Size of the Information Being Gathered

Use the minimal option (-m):

```
supportconfig -m
```

Limiting the Information to a Specific Topic

If you have already localized a problem with the default **supportconfig** output and have found that it relates to a specific area or feature set only, you should limit the collected information to the specific area for the next **supportconfig** run. For example, if you detected problems with LVM and want to test a recent change that you did to the LVM configuration, it makes sense to gather the minimum supportconfig information around LVM only:

```
supportconfig -i LVM
```

For a complete list of feature keywords that you can use for limiting the collected information to a specific area, run

```
supportconfig -F
```

Including Additional Contact Information in the Output:

```
supportconfig -E tux@example.org -N "Tux Penguin" -O "Penguin Inc." ...
```

(all in one line)

Collecting Already Rotated Log Files

```
supportconfig -l
```

This is especially useful in high logging environments or after a Kernel crash when syslog rotates the log files after a reboot.

37.3 Submitting Information to Global Technical Support

Use the YaST *Support* module or the **supportconfig** command line utility to submit system information to the Global Technical Support. When you experience a server issue and want the support's assistance, you will need to open a service request first. For details, see [Section 37.2.1, "Creating a Service Request Number"](#).

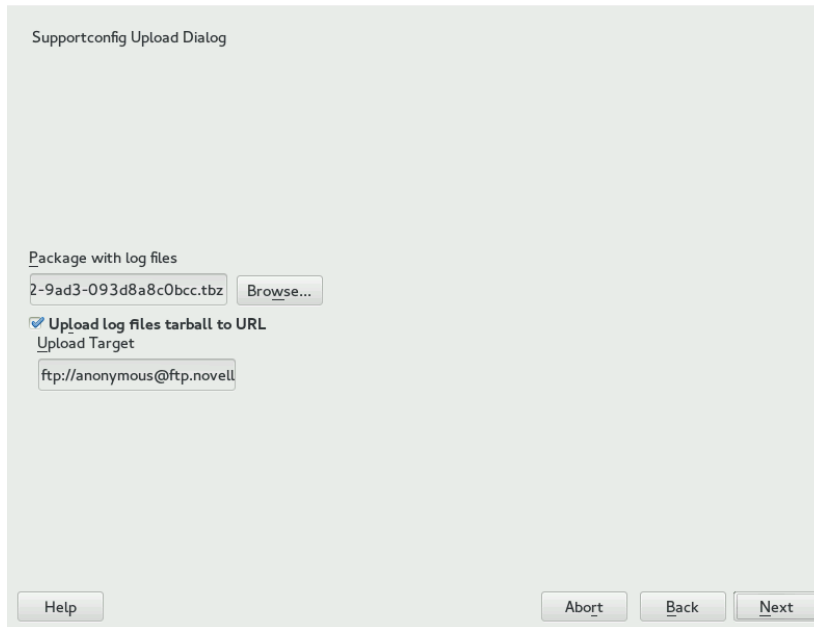
The following examples use 12345678901 as a placeholder for your service request number. Replace 12345678901 with the service request number you created in [Section 37.2.1, "Creating a Service Request Number"](#).

PROCEDURE 37.1: SUBMITTING INFORMATION TO SUPPORT WITH YAST

The following procedure assumes that you have already created a supportconfig archive, but have not uploaded it yet. Make sure to have included your contact information in the archive as described in [Section 37.2.3, "Creating a Supportconfig Archive with YaST", Step 4](#). For instructions on how to generate and submit a supportconfig archive in one go, see [Section 37.2.3, "Creating a Supportconfig Archive with YaST"](#).

1. Start YaST and open the *Support* module.

2. Click *Upload*.
3. In *Package with log files* specify the path to the existing supportconfig archive or *Browse* for it.
4. YaST automatically proposes an upload server. If you want to modify it, refer to [Section 37.2.2, “Upload Targets”](#) for details of which upload servers are available.



Proceed with *Next*.

5. Click *Finish*.

PROCEDURE 37.2: SUBMITTING INFORMATION TO SUPPORT FROM COMMAND LINE

The following procedure assumes that you have already created a supportconfig archive, but have not uploaded it yet. For instructions on how to generate and submit a supportconfig archive in one go, see [Section 37.2.3, “Creating a Supportconfig Archive with YaST”](#).

1. Servers with Internet connectivity:
 - a. To use the default upload target, run:

```
supportconfig -ur 12345678901
```

- b. For the secure upload target, use the following:

```
supportconfig -ar 12345678901
```

2. Servers *without* Internet connectivity

a. Run the following:

```
supportconfig -r 12345678901
```

b. Manually upload the `/var/log/nts_SR12345678901*tbz` archive to one of our FTP servers. Which one to use depends on your location in the world. For an overview, see [Section 37.2.2, “Upload Targets”](#).

3. After the TAR archive arrives in the incoming directory of our FTP server, it becomes automatically attached to your service request.

37.4 Analyzing System Information

System reports created with **supportconfig** can be analyzed for known issues to help resolve problems faster. For this purpose, SUSE Linux Enterprise Server provides both an appliance and a command line tool for Supportconfig Analysis (SCA). The SCA appliance is a server-side tool which is non-interactive. The SCA tool (**scatool**) runs on the client-side and is executed from command line. Both tools analyze supportconfig archives from affected servers. The initial server analysis takes place on the SCA appliance or the workstation on which scatool is running. No analysis cycles happen on the production server.

Both the appliance and the command line tool additionally need product-specific patterns that enable them to analyze the supportconfig output for the associated products. Each pattern is a script that parses and evaluates a supportconfig archive for one known issue. The patterns are available as RPM packages.

For example, if you want to analyze supportconfig archives that have been generated on a SUSE Linux Enterprise 11 machine, you need to install the package `sca-patterns-sle11` together with the SCA tool (or on the machine that you want to use as the SCA appliance server). To analyze supportconfig archives generated on a SUSE Linux Enterprise 10 machine, you need the package `sca-patterns-sle10`.

You can also develop your own patterns as briefly described in [Section 37.4.3, “Developing Custom Analysis Patterns”](#).

37.4.1 SCA Command Line Tool

The SCA command line tool lets you analyze a local machine using both **supportconfig** and the analysis patterns for the specific product that is installed on the local machine. The tool creates an HTML report showing its analysis results. For an example, see *Figure 37.1, “HTML Report Generated by SCA Tool”*.

Supportconfig Analysis Report

Server Information

Analysis Date:
Archive File:

/4/25/2014 11:22
/var/log/nts_barett-2_140425_1119.html

Server Name: barett-2
Distribution: SUSE Linux Enterprise Server 12 (x86_64)
Hypervisor: KVM (QEMU Virtual CPU)
Kernel Version: 3.12.14-1-default

Hardware: Bochs
Service Pack: 0
Identity: Virtual Machine (QEMU Virtual CPU)
Supportconfig Version: 3.0-18

Conditions Evaluated as Critical

Category	Message	Solutions
Basic Health	2 Basic Health Message(s)	
Basic Health SLE	Kernel Kernel Status -- Tainted: F O	TID
Basic Health SLE	System Last system down was not clean on Mon Mar 24 17:37:04 2014 and 1 additional failure(s)	TID TID1
SLE	2 SLE Message(s)	

Conditions Evaluated as Warning

Category	Message	Solutions
SLE	1 SLE Message(s)	

Conditions Evaluated as Recommended

Category	Message	Solutions
SLE	1 SLE Message(s)	

Conditions Evaluated as Success

Category	Message	Solutions
Security	1 Security Message(s)	
Security SLE	AppArmor There are no AppArmor reject messages	TID Doc
Basic Health	8 Basic Health Message(s)	
Basic Health SLE	Kernel Context switches per second observed: 79	TID
Basic Health SLE	Kernel Interrupts per second observed: 51	TID
Basic Health SLE	CPU Utilization: 1.00%, Idle: 99.00%	TID
Basic Health SLE	Disk Mount on / has highest used space: 22%	TID TID2
Basic Health SLE	Kernel 2% CPU load within limits, CPUs: 1, Load Average: 0.02	TID Web Wikipedia
Basic Health SLE	Memory Memory used 29% - Swapping: No	TID
Basic Health SLE	Processes 0 Uninterruptible processes observed	TID
Basic Health SLE	Processes 0 Zombie processes observed	TID

FIGURE 37.1: HTML REPORT GENERATED BY SCA TOOL

The **scatool** command is provided by the **sca-server-report** package. It is not installed by default. Additionally, you need the **sca-patterns-base** package and any of the product-specific **sca-patterns-*** packages that matches the product installed on the machine where you want to run the **scatool** command.

Execute the **scatool** command either as **root** user or with **sudo**. When calling the SCA tool, you can either analyze an existing **supportconfig** TAR archive or you can let it generate and analyze a new archive in one go. The tool also provides an interactive console (with tab completion) and the possibility to run **supportconfig** on an external machine and to execute the subsequent analysis on the local machine.

Find some example commands below:

sudo scatool -s

Calls **supportconfig** and generates a new supportconfig archive on the local machine. Analyzes the archive for known issues by applying the SCA analysis patterns that match the installed product. Displays the path to the HTML report that is generated from the results of the analysis. It is usually written to the same directory where the supportconfig archive can be found.

sudo scatool -s -o /opt/sca/reports/

Same as **sudo scatool -s**, only that the HTML report is written to the path specified with **-o**.

sudo scatool -a PATH_TO_TARBALL_OR_DIR

Analyzes the specified supportconfig archive file (or the specified directory to where the supportconfig archive has been extracted). The generated HTML report is saved in the same location as the supportconfig archive or directory.

sudo scatool -a sles_server.company.com

Establishes an SSH connection to an external server **sles_server.company.com** and runs **supportconfig** on the server. The supportconfig archive is then copied back to the local machine and is analyzed there. The generated HTML report is saved to the default **/var/log** directory. (Only the supportconfig archive is created on **sles_server.company.com**).

sudo scatool -c

Starts the interactive console for **scatool**. Press  twice to see the available commands.

For further options and information, run **sudo scatool -h** or see the **scatool** man page.

37.4.2 SCA Appliance

If you decide to use the SCA appliance for analyzing the supportconfig archives, you need to configure a dedicated server (or virtual machine) as the SCA appliance server. The SCA appliance server can then be used to analyze supportconfig archives from all machines in your enterprise running SUSE Linux Enterprise Server or SUSE Linux Enterprise Desktop. You can simply upload supportconfig archives to the appliance server for analysis. Interaction is not required. In a MariaDB database, the SCA appliance keeps track of all supportconfig archives that have been

analyzed . You can read the SCA reports directly from the appliance Web interface. Alternatively, you can have the appliance send the HTML report to any administrative user via e-mail. For details, see [Section 37.4.2.5.4, “Sending SCA Reports via E-Mail”](#).

37.4.2.1 Installation Quick Start

To install and set up the SCA appliance in a very fast way from the command line, follow the instructions here. The procedure is intended for experts and focuses on the bare installation and setup commands. For more information, refer to the more detailed description in [Section 37.4.2.2, “Prerequisites”](#) to [Section 37.4.2.3, “Installation and Basic Setup”](#).

PREREQUISITES

- Web and LAMP Pattern
- Web and Scripting Module (you must register the machine to be able to select this module).



Note: root Privileges Required

All commands in the following procedure must be run as root.

PROCEDURE 37.3: INSTALLATION USING ANONYMOUS FTP FOR UPLOAD

After the appliance is set up and running, no more manual interaction is required. This way of setting up the appliance is therefore ideal for using cron jobs to create and upload supportconfig archives.

1. On the machine on which to install the appliance, log in to a console and execute the following commands:

```
zypper install sca-appliance-* sca-patterns-* vsftpd
systemctl enable apache2
systemctl start apache2
systemctl enable vsftpd
systemctl start vsftpd
yast ftp-server
```

2. In YaST FTP Server, select *Authentication > Enable Upload > Anonymous Can Upload > Finish > Yes to Create /srv/ftp/upload*.

3. Execute the following commands:

```
systemctl enable mysql
systemctl start mysql
mysql_secure_installation
setup-sca -f
```

The `mysql_secure_installation` will create a MariaDB `root` password.

PROCEDURE 37.4: INSTALLATION USING SCP/TMP FOR UPLOAD

This way of setting up the appliance requires manual interaction when typing the SSH password.

1. On the machine on which to install the appliance, log in to a console.
2. Execute the following commands:

```
zypper install sca-appliance-* sca-patterns-*
systemctl enable apache2
systemctl start apache2
sudo systemctl enable mysql
systemctl start mysql
mysql_secure_installation
setup-sca
```

37.4.2.2 Prerequisites

To run an SCA appliance server, you need the following prerequisites:

- All `sca-appliance-*` packages.
- The `sca-patterns-base` package. Additionally, any of the product-specific `sca-patterns-*` for the type of supportconfig archives that you want to analyze with the appliance.
- Apache
- PHP
- MariaDB
- anonymous FTP server (optional)

37.4.2.3 Installation and Basic Setup

As listed in [Section 37.4.2.2, “Prerequisites”](#), the SCA appliance has several dependencies on other packages. Therefore you need do some preparations before installing and setting up the SCA appliance server:

1. For Apache and MariaDB, install the [Web](#) and [LAMP](#) installation patterns.
2. Set up Apache, MariaDB, and optionally an anonymous FTP server. For more information, see [Chapter 29, The Apache HTTP Server](#) and [Chapter 30, Setting Up an FTP Server with YaST](#).
3. Configure Apache and MariaDB to start at boot time:

```
sudo systemctl enable apache2 mysql
```

4. Start both services:

```
sudo systemctl start apache2 mysql
```

Now you can install the SCA appliance and set it up as described in [Procedure 37.5, “Installing and Configuring the SCA Appliance”](#).

PROCEDURE 37.5: INSTALLING AND CONFIGURING THE SCA APPLIANCE

After installing the packages, use the [setup-sca](#) script for the basic configuration of the MariaDB administration and report database that is used by the SCA appliance.

It can be used to configure the following options you have for uploading the supportconfig archives from your machines to the SCA appliance:

- [scp](#)
- anonymous FTP server

1. Install the appliance and the SCA base-pattern library:

```
sudo zypper install sca-appliance-* sca-patterns-base
```

2. Additionally, install the pattern packages for the types of supportconfig archives you want to analyze. For example, if you have SUSE Linux Enterprise Server 11 and SUSE Linux Enterprise Server 12 servers in your environment, install both the [sca-patterns-sle11](#) and [sca-patterns-sle12](#) packages.

To install all available patterns:

```
zypper install sca-patterns-*
```

3. For basic setup of the SCA appliance, use the **setup-sca** script. How to call it depends on how you want to upload the supportconfig archives to the SCA appliance server:

- If you have configured an anonymous FTP server that uses the /srv/ftp/upload directory, execute the setup script with the -f option and follow the instructions on the screen:

```
setup-sca -f
```



Note: FTP Server Using Another Directory

If your FTP server uses another directory than /srv/ftp/upload, adjust the following configuration files first to make them point to the correct directory: /etc/sca/sdagent.conf and /etc/sca/sdbroker.conf.

- If you want to upload supportconfig files to the /tmp directory of the SCA appliance server via **scp**, call the setup script without any parameters and follow the instructions on the screen:

```
setup-sca
```

The setup script runs a few checks regarding its requirements and configures the needed components. It will prompt you for two passwords: the MySQL root password of the MariaDB that you have set up, and a Web user password with which to log in to the Web interface of the SCA appliance.

4. Enter the existing MariaDB root password. It will allow the SCA appliance to connect to the MariaDB.
5. Define a password for the Web user. It will be written to /srv/www/htdocs/sca/web-config.php and will be set as the password for the user scdiag. Both user name and password can be changed at any time later, see *Section 37.4.2.5.1, "Password for the Web Interface"*.

After successful installation and setup, the SCA appliance is ready for use, see [Section 37.4.2.4, “Using the SCA Appliance”](#). However, you should modify some options such as changing the password for the Web interface, changing the source for the SCA pattern updates, enabling archiving mode or configuring e-mail notifications. For details on that, see [Section 37.4.2.5, “Customizing the SCA Appliance”](#).



Warning: Data Protection

As the reports on the SCA appliance server contain security-relevant information of the machines whose supportconfig archives have been analyzed, make sure to protect the data on the SCA appliance server against unauthorized access.

37.4.2.4 Using the SCA Appliance

You can upload existing supportconfig archives to the SCA appliance manually or create new supportconfig archives and upload them to the SCA appliance in one step. Uploading can be done via FTP or SCP. For both, you need to know the URL where the SCA appliance can be reached. For upload via FTP, an FTP server needs to be configured for the SCA appliance, see [Procedure 37.5, “Installing and Configuring the SCA Appliance”](#).

37.4.2.4.1 Uploading Supportconfig Archives to the SCA Appliance

- For creating a supportconfig archive and uploading it via (anonymous) FTP:

```
sudo supportconfig -U "ftp://sca-appliance.company.com/upload"
```

- For creating a supportconfig archive and uploading it via SCP:

```
sudo supportconfig -U "scp://sca-appliance.company.com/tmp"
```

You will be prompted for the root user password of the server running the SCA appliance.

- If you want to manually upload one or multiple archives, copy the existing archive files (usually located at /var/log/nts_*.tbz) to the SCA appliance. As target, use either the appliance server's /tmp directory or the /srv/ftp/upload directory (if FTP is configured for the SCA appliance server).

37.4.2.4.2 Viewing SCA Reports

SCA reports can be viewed from any machine that has a browser installed and can access the report index page of the SCA appliance.

1. Start a Web browser and make sure that JavaScript and cookies are enabled.
2. As a URL, enter the report index page of the SCA appliance.

`https://sca-appliance.company.com/sca`

If in doubt, ask your system administrator.

3. You will be prompted for a user name and a password to log in.

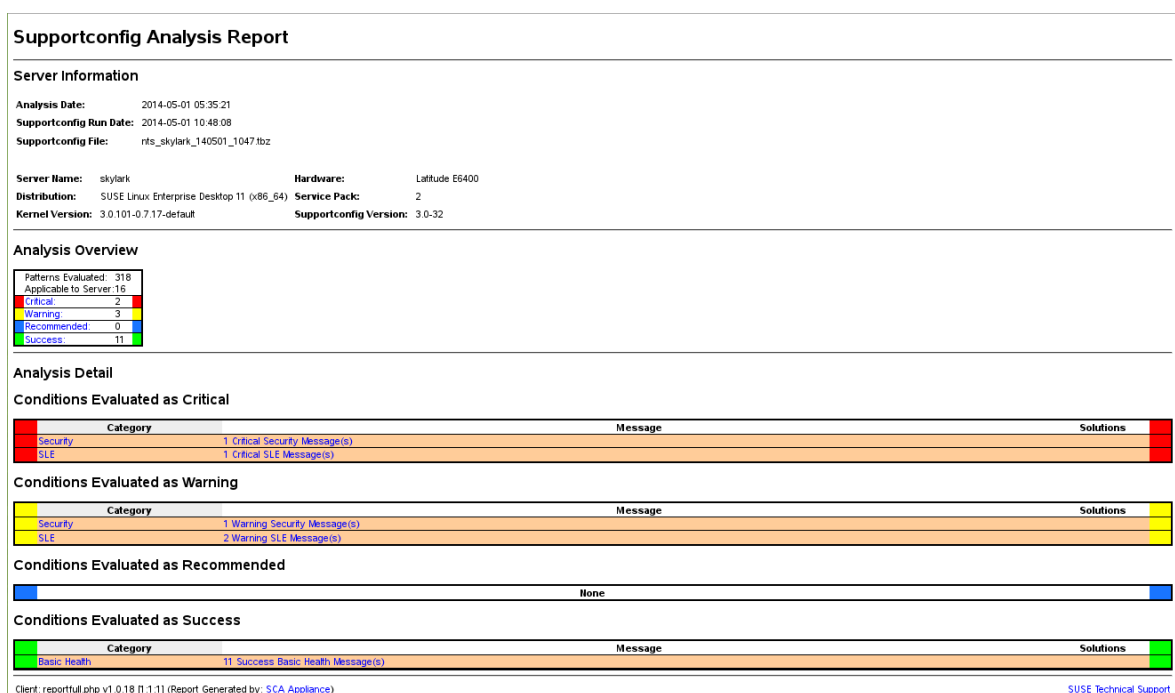


FIGURE 37.2: HTML REPORT GENERATED BY SCA APPLIANCE

4. After logging in, click the date of the report you want to read.
5. Click the *Basic Health* category first to expand it.
6. In the *Message* column, click an individual entry. This opens the corresponding article in the SUSE Knowledgebase. Read the proposed solution and follow the instructions.
7. If the *Solutions* column of the *Supportconfig Analysis Report* shows any additional entries, click them. Read the proposed solution and follow the instructions.

8. Check the SUSE Knowledgebase (<http://www.suse.com/support/kb/>) for results that directly relate to the problem identified by SCA. Work at resolving them.
9. Check for results that can be addressed proactively to avoid future problems.

37.4.2.5 Customizing the SCA Appliance

The following sections show how to change the password for the Web interface, how to change the source for the SCA pattern updates, how to enable archiving mode, and how to configure e-mail notifications.

37.4.2.5.1 Password for the Web Interface

The SCA Appliance Web interface requires a user name and password for logging in. The default user name is `scdiag` and the default password is `linux` (if not specified otherwise, see *Procedure 37.5, "Installing and Configuring the SCA Appliance"*). Change the default password to a secure password at the earliest possibility. You can also modify the user name.

PROCEDURE 37.6: CHANGING USER NAME OR PASSWORD FOR THE WEB INTERFACE

1. Log in as `root` user at the system console of the SCA appliance server.
2. Open `/srv/www/htdocs/sca/web-config.php` in an editor.
3. Change the values of `$username` and `$password` as desired.
4. Save the file and exit.

37.4.2.5.2 Updates of SCA Patterns

By default, all `sca-patterns-*` packages are updated regularly by a `root` cron job that executes the `sdagent-patterns` script nightly, which in turn runs `zypper update sca-patterns-*`. A regular system update will update all SCA appliance and pattern packages. To update the SCA appliance and patterns manually, run:

```
sudo zypper update sca-*
```

The updates are installed from the SUSE Linux Enterprise 12 SP2 update repository by default. You can change the source for the updates to an SMT server, if desired. When `sdagent-patterns` runs `zypper update sca-patterns-*`, it gets the updates from the currently configured update channel. If that channel is located on an SMT server, the packages will be pulled from there.

PROCEDURE 37.7: **DISABLING AUTOMATIC UPDATES OF SCA PATTERNS**

1. Log in as `root` user at the system console of the SCA appliance server.
2. Open `/etc/sca/sdagent-patterns.conf` in an editor.
3. Change the entry

```
UPDATE_FROM_PATTERN_REPO=1
```

to

```
UPDATE_FROM_PATTERN_REPO=0
```

4. Save the file and exit. The machine does not require any restart to apply the change.

37.4.2.5.3 **Archiving Mode**

All supportconfig archives are deleted from the SCA appliance after they have been analyzed and their results have been stored in the MariaDB database. However, for troubleshooting purposes it can be useful to keep copies of supportconfig archives from a machine. By default, archiving mode is disabled.

PROCEDURE 37.8: **ENABLING ARCHIVING MODE IN THE SCA APPLIANCE**

1. Log in as `root` user at the system console of the SCA appliance server.
2. Open `/etc/sca/sdagent.conf` in an editor.
3. Change the entry

```
ARCHIVE_MODE=0
```

to

```
ARCHIVE_MODE=1
```

4. Save the file and exit. The machine does not require any restart to apply the change.

After having enabled archive mode, the SCA appliance will save the supportconfig files to the /var/log/archives/saved directory, instead of deleting them.

37.4.2.5.4 Sending SCA Reports via E-Mail

The SCA appliance can e-mail a report HTML file for each supportconfig analyzed. This feature is disabled by default. When enabling it, you can define a list of e-mail addresses to which the reports should be sent, and define a level of status messages that trigger the sending of reports (STATUS_NOTIFY_LEVEL).

POSSIBLE VALUES FOR STATUS_NOTIFY_LEVEL

\$STATUS_OFF

Deactivate sending of HTML reports.

\$STATUS_CRITICAL

Send only SCA reports that include a CRITICAL.

\$STATUS_WARNING

Send only SCA reports that include a WARNING or CRITICAL.

\$STATUS_RECOMMEND

Send only SCA reports that include a RECOMMEND, WARNING or CRITICAL.

\$STATUS_SUCCESS

Send SCA reports that include a SUCCESS, RECOMMEND, WARNING or CRITICAL.

PROCEDURE 37.9: CONFIGURING E-MAIL NOTIFICATIONS FOR SCA REPORTS

1. Log in as root user at the system console of the SCA appliance server.
2. Open /etc/sca/sdagent.conf in an editor.
3. Search for the entry STATUS_NOTIFY_LEVEL. By default, it is set to \$STATUS_OFF (e-mail notifications are disabled).
4. To enable e-mail notifications, change \$STATUS_OFF to the level of status messages that you want to have e-mail reports for, for example:

```
STATUS_NOTIFY_LEVEL=$STATUS_SUCCESS
```


For details, see *Possible Values for STATUS_NOTIFY_LEVEL*.

5. To define the list of recipients to which the reports should be sent:
 - a. Search for the entry `EMAIL_REPORT='root'`.
 - b. Replace `root` with a list of e-mail addresses to which SCA reports should be sent. The e-mail addresses must be separated by spaces. For example:

```
EMAIL_REPORT='tux@my.company.com wilber@your.company.com'
```
6. Save the file and exit. The machine does not require any restart to apply the changes. All future SCA reports will be e-mailed to the specified addresses.

37.4.2.6 Backing Up and Restoring the Database

To back up and restore the MariaDB database that stores the SCA reports, use the `scadb` command as described below.

PROCEDURE 37.10: BACKING UP THE DATABASE

1. Log in as `root` user at the system console of the server running the SCA appliance.
2. Put the appliance into maintenance mode by executing:

```
scadb maint
```

3. Start the backup with:

```
scadb backup
```

The data is saved to a TAR archive: `sca-backup-*.sql.gz`.

4. If you are using the pattern creation database to develop your own patterns (see *Section 37.4.3, "Developing Custom Analysis Patterns"*), back up this data, too:

```
sdpdb backup
```

The data is saved to a TAR archive: `sdp-backup-*.sql.gz`.

5. Copy the following data to another machine or an external storage medium:

- sca-backup-*.sql.gz
- sdp-backup-*.sql.gz
- /usr/lib/sca/patterns/local (only needed if you have created custom patterns)

6. Reactivate the SCA appliance with:

```
scadb reset agents
```

PROCEDURE 37.11: RESTORING THE DATABASE

To restore the database from your backup, proceed as follows:

1. Log in as root user at the system console of the server running the SCA appliance.
2. Copy the newest sca-backup-*.sql.gz and sdp-backup-*.sql.gz TAR archives to the SCA appliance server.
3. To decompress the files, run:

```
gzip -d *-backup-*.sql.gz
```

4. To import the data into the database, execute:

```
scadb import sca-backup-*.sql
```

5. If you are using the pattern creation database to create your own patterns, also import the following data with:

```
sdpdb import sdp-backup-*.sql
```

6. If you are using custom patterns, also restore /usr/lib/sca/patterns/local from your backup data.

7. Reactivate the SCA appliance with:

```
scadb reset agents
```

8. Update the pattern modules in the database with:

```
sdagent-patterns -u
```

37.4.3 Developing Custom Analysis Patterns

The SCA appliance comes with a complete pattern development environment (the SCA Pattern Database) that enables you to develop your own, custom patterns. Patterns can be written in any programming language. To make them available for the `supportconfig` analysis process, they need to be saved to `/usr/lib/sca/patterns/local` and to be made executable. Both the SCA appliance and the SCA tool will then run the custom patterns against new `supportconfig` archives as part of the analysis report. For detailed instructions on how to create (and test) your own patterns, see <http://www.suse.com/communities/conversations/sca-pattern-development/>.

37.5 Gathering Information During the Installation

During the installation, `supportconfig` is not available. However, you can collect log files from YaST by using `save_y2logs`. This command will create a `.tar.xz` archive in the directory `/tmp`.

If issues appear very early during installation, you may be able to gather information from the log file created by `linuxrc`. `linuxrc` is a small command that runs before YaST starts. This log file is available at `/var/log/linuxrc.log`.



Important: Installation Log Files Not Available in the Installed System

The log files available during the installation are not available in the installed system anymore. Properly save the installation log files while the installer is still running.

37.6 Support of Kernel Modules

An important requirement for every enterprise operating system is the level of support you receive for your environment. Kernel modules are the most relevant connector between hardware (“controllers”) and the operating system. Every Kernel module in SUSE Linux Enterprise has a supported flag that can take three possible values:

- “yes”, thus supported
- “external”, thus supported
- “” (empty, not set), thus unsupported

The following rules apply:

- All modules of a self-recompiled Kernel are by default marked as unsupported.
- Kernel modules supported by SUSE partners and delivered using SUSE SolidDriver Program are marked “external”.
- If the supported flag is not set, loading this module will taint the Kernel. Tainted Kernels are not supported. Unsupported Kernel modules are included in an extra RPM package (kernel-FLAVOR-extra) that is only available for SUSE Linux Enterprise Desktop and the SUSE Linux Enterprise Workstation Extension. Those kernels will not be loaded by default (FLAVOR = default | xen | ...). In addition, these unsupported modules are not available in the installer, and the kernel-FLAVOR-extra package is not part of the SUSE Linux Enterprise media.
- Kernel modules not provided under a license compatible to the license of the Linux Kernel will also taint the Kernel. For details, see /usr/src/linux/Documentation/sysctl/kernel.txt and the state of /proc/sys/kernel/tainted.

37.6.1 Technical Background

- **Linux Kernel:** The value of `/proc/sys/kernel/unsupported` defaults to `2` on SUSE Linux Enterprise 12 SP2 (`do not warn in syslog when loading unsupported modules`). This default is used in the installer and in the installed system. See `/usr/src/linux/Documentation/sysctl/kernel.txt` for more information.
- **modprobe:** The `modprobe` utility for checking module dependencies and loading modules appropriately checks for the value of the `supported` flag. If the value is “yes” or “external” the module will be loaded, otherwise it will not. For information on how to override this behavior, see *Section 37.6.2, “Working with Unsupported Modules”*.



Note: Support

SUSE does not generally support the removal of storage modules via `modprobe -r`.





37.6.2 Working with Unsupported Modules

While general supportability is important, situations can occur where loading an unsupported module is required (for example, for testing or debugging purposes, or if your hardware vendor provides a hotfix).

- To override the default, edit `/etc/modprobe.d/10-unsupported-modules.conf` and change the value of the variable `allow_unsupported_modules` to `1`. If an unsupported module is needed in the initrd, do not forget to run `dracut -f` to update the initrd. If you only want to try loading a module once, you can use the `--allow-unsupported-modules` option with `modprobe`. For more information, see the `modprobe` man page.
- During installation, unsupported modules may be added through driver update disks, and they will be loaded. To enforce loading of unsupported modules during boot and afterward, use the Kernel command line option `oem-modules`. While installing and initializing the `suse-module-tools` package, the Kernel flag `TAINT_NO_SUPPORT` (`/proc/sys/kernel/tainted`) will be evaluated. If the Kernel is already tainted, `allow_unsupported_modules` will be enabled. This will prevent unsupported modules from failing in the system being installed. If no unsupported modules are present during installation and the other special Kernel command line option (`oem-modules=1`) is not used, the default still is to disallow unsupported modules.

Remember that loading and running unsupported modules will make the Kernel and the whole system unsupported by SUSE.

37.7 For More Information

- **`man supportconfig`**—The **`supportconfig`** man page.
- **`man supportconfig.conf`**—The man page of the supportconfig configuration file.
- **`man scatool`**—The **`scatool`** man page.
- **`man scadb`**—The **`scadb`** man page.
- **`man setup-sca`**—The **`setup-sca`** man page.
- <https://mariadb.com/kb/en/> —The MariaDB documentation.
- <http://httpd.apache.org/docs/>  and *Chapter 29, The Apache HTTP Server*—Documentation about the Apache Web server.
- *Chapter 30, Setting Up an FTP Server with YaST*—Documentation of how to set up an FTP server.
- <http://www.suse.com/communities/conversations/sca-pattern-development/> —Instructions on how to create (and test) your own SCA patterns.
- <http://www.suse.com/communities/conversations/basic-server-health-check-supportconfig/> —A Basic Server Health Check with Supportconfig.
- https://www.novell.com/communities/cooltools/cool_tools/create-your-own-supportconfig-plugin/ —Create Your Own Supportconfig Plugin.
- <http://www.suse.com/communities/conversations/creating-a-central-supportconfig-repository/> —Creating a Central Supportconfig Repository.

38 Common Problems and Their Solutions

This chapter describes a range of potential problems and their solutions. Even if your situation is not precisely listed here, there may be one similar enough to offer hints to the solution of your problem.

38.1 Finding and Gathering Information

Linux reports things in a very detailed way. There are several places to look when you encounter problems with your system, most of which are standard to Linux systems in general, and some are relevant to SUSE Linux Enterprise Server systems. Most log files can be viewed with YaST (*Miscellaneous > Start-Up Log*).

YaST offers the possibility to collect all system information needed by the support team. Use *Other > Support* and select the problem category. When all information is gathered, attach it to your support request.

A list of the most frequently checked log files follows with the description of their typical purpose. Paths containing `~` refer to the current user's home directory.

TABLE 38.1: LOG FILES

Log File	Description
<u><code>~/.xsession-errors</code></u>	Messages from the desktop applications currently running.
<u><code>/var/log/apparmor/</code></u>	Log files from AppArmor, see <i>Book “Security Guide”</i> for detailed information.
<u><code>/var/log/audit/audit.log</code></u>	Log file from Audit to track any access to files, directories, or resources of your system, and trace system calls. See <i>Book “Security Guide”</i> for detailed information.
<u><code>/var/log/mail.*</code></u>	Messages from the mail system.
<u><code>/var/log/NetworkManager</code></u>	Log file from NetworkManager to collect problems with network connectivity

Log File	Description
<u>/var/log/samba/</u>	Directory containing Samba server and client log messages.
<u>/var/log/warn</u>	All messages from the kernel and system log daemon with the “warning” level or higher.
<u>/var/log/wtmp</u>	Binary file containing user login records for the current machine session. View it with <u>last</u> .
<u>/var/log/Xorg.*.log</u>	Various start-up and runtime log files from the X Window System. It is useful for debugging failed X start-ups.
<u>/var/log/YaST2/</u>	Directory containing YaST's actions and their results.
<u>/var/log/zypper.log</u>	Log file of Zypper.

Apart from log files, your machine also supplies you with information about the running system. See *Table 38.2: System Information With the /proc File System*

TABLE 38.2: SYSTEM INFORMATION WITH THE /proc FILE SYSTEM

File	Description
<u>/proc/cpuinfo</u>	Contains processor information, including its type, make, model, and performance.
<u>/proc/dma</u>	Shows which DMA channels are currently being used.
<u>/proc/interrupts</u>	Shows which interrupts are in use, and how many of each have been in use.
<u>/proc/iomem</u>	Displays the status of I/O (input/output) memory.

File	Description
<u>/proc/ioports</u>	Shows which I/O ports are in use at the moment.
<u>/proc/meminfo</u>	Displays memory status.
<u>/proc/modules</u>	Displays the individual modules.
<u>/proc/mounts</u>	Displays devices currently mounted.
<u>/proc/partitions</u>	Shows the partitioning of all hard disks.
<u>/proc/version</u>	Displays the current version of Linux.

Apart from the /proc file system, the Linux kernel exports information with the sysfs module, an in-memory file system. This module represents kernel objects, their attributes and relationships. For more information about sysfs, see the context of udev in *Chapter 19, Dynamic Kernel Device Management with udev*. *Table 38.3* contains an overview of the most common directories under /sys.

TABLE 38.3: SYSTEM INFORMATION WITH THE /sys FILE SYSTEM

File	Description
<u>/sys/block</u>	Contains subdirectories for each block device discovered in the system. Generally, these are mostly disk type devices.
<u>/sys/bus</u>	Contains subdirectories for each physical bus type.
<u>/sys/class</u>	Contains subdirectories grouped together as a functional types of devices (like graphics, net, printer, etc.)
<u>/sys/device</u>	Contains the global device hierarchy.

Linux comes with several tools for system analysis and monitoring. See *Book “System Analysis and Tuning Guide”, Chapter 2 “System Monitoring Utilities”* for a selection of the most important ones used in system diagnostics.

Each of the following scenarios begins with a header describing the problem followed by a paragraph or two offering suggested solutions, available references for more detailed solutions, and cross-references to other scenarios that are related.

38.2 Installation Problems

Installation problems are situations when a machine fails to install. It may fail entirely or it may not be able to start the graphical installer. This section highlights some typical problems you may run into, and offers possible solutions or workarounds for these kinds of situations.

38.2.1 Checking Media

If you encounter any problems using the SUSE Linux Enterprise Server installation media, check the integrity of your installation media. Boot from the media and choose *Check Installation Media* from the boot menu. In a running system, start YaST and choose *Software > Media Check*. To check the SUSE Linux Enterprise Server medium, insert it into the drive and click *Start Check* in the *Media Check* screen of YaST. This may take several minutes. If errors are detected, do not use this medium for installation. Media problems may occur when having burned the medium yourself. Burning the media at a low speed (4x) helps to avoid problems.

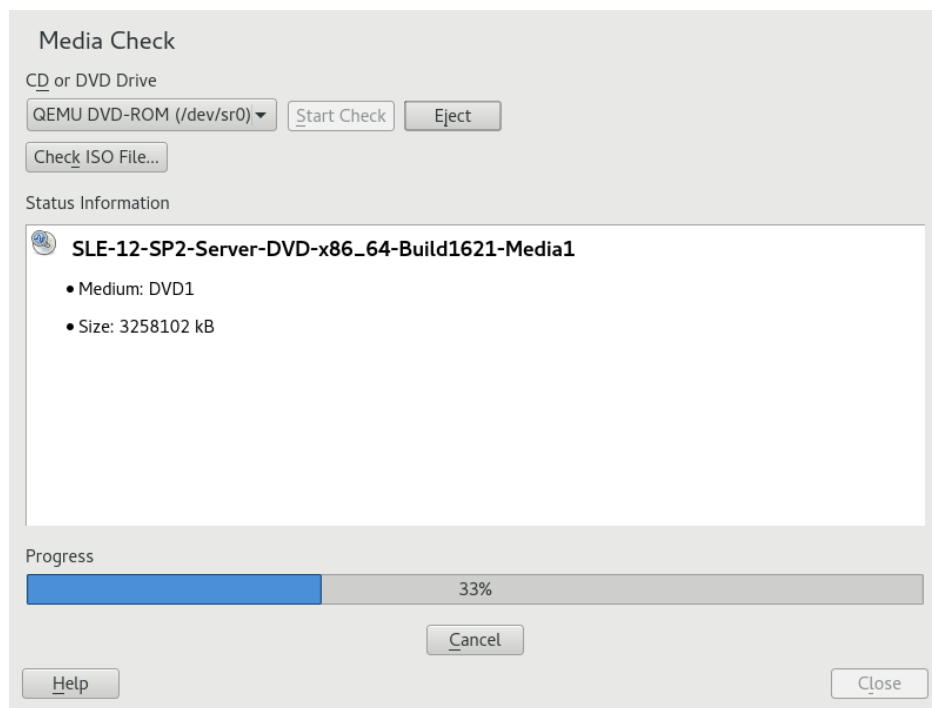


FIGURE 38.1: CHECKING MEDIA

38.2.2 No Bootable DVD Drive Available

If your computer does not contain a bootable DVD-ROM drive or if the one you have is not supported by Linux, there are several options you can install your machine without a built-in DVD drive:

Using an External Boot Device

If it is supported by your BIOS and the installation kernel, boot from external DVD drives or USB storage devices. Refer to *Book “Deployment Guide”, Chapter 6 “Installation with YaST”, Section 6.2.2 “PC (AMD64/Intel 64/ARM AArch64): System Start-up”* for instructions on how to create a bootable USB storage device.

Network Boot via PXE

If a machine lacks a DVD drive, but provides a working Ethernet connection, perform a completely network-based installation. See *Book “Deployment Guide”, Chapter 9 “Remote Installation”, Section 9.1.3 “Remote Installation via VNC—PXE Boot and Wake on LAN”* and *Book “Deployment Guide”, Chapter 9 “Remote Installation”, Section 9.1.6 “Remote Installation via SSH—PXE Boot and Wake on LAN”* for details.

38.2.2.1 External Boot Devices

Linux supports most existing DVD drives. If the system has no DVD drive, it is still possible that an external DVD drive, connected through USB, FireWire, or SCSI, can be used to boot the system. This depends mainly on the interaction of the BIOS and the hardware used. Sometimes a BIOS update may help if you encounter problems.

When installing from a Live CD, you can also create a “Live flash disk” to boot from.

38.2.3 Booting from Installation Media Fails

One reason a machine does not boot the installation media can be an incorrect boot sequence setting in BIOS. The BIOS boot sequence must have DVD drive set as the first entry for booting. Otherwise the machine would try to boot from another medium, typically the hard disk. Guidance for changing the BIOS boot sequence can be found in the documentation provided with your mainboard, or in the following paragraphs.

The BIOS is the software that enables the very basic functions of a computer. Motherboard vendors provide a BIOS specifically made for their hardware. Normally, the BIOS setup can only be accessed at a specific time—when the machine is booting. During this initialization phase, the machine performs several diagnostic hardware tests. One of them is a memory check, indicated by a memory counter. When the counter appears, look for a line, usually below the counter or somewhere at the bottom, mentioning the key to press to access the BIOS setup. Usually the key to press is one of `[Del]`, `[F1]`, or `[Esc]`. Press this key until the BIOS setup screen appears.

PROCEDURE 38.1: CHANGING THE BIOS BOOT SEQUENCE

1. Enter the BIOS using the proper key as announced by the boot routines and wait for the BIOS screen to appear.
2. To change the boot sequence in an AWARD BIOS, look for the *BIOS FEATURES SETUP* entry. Other manufacturers may have a different name for this, such as *ADVANCED CMOS SETUP*. When you have found the entry, select it and confirm with `[Enter]`.
3. In the screen that opens, look for a subentry called *BOOT SEQUENCE* or *BOOT ORDER*. Change the settings by pressing `[Page ↑]` or `[Page ↓]` until the DVD drive is listed first.
4. Leave the BIOS setup screen by pressing `[Esc]`. To save the changes, select *SAVE & EXIT SETUP*, or press `[F10]`. To confirm that your settings should be saved, press `[Y]`.

PROCEDURE 38.2: CHANGING THE BOOT SEQUENCE IN AN SCSI BIOS (ADAPTEC HOST ADAPTER)

1. Open the setup by pressing **Ctrl**–**A**.
2. Select *Disk Utilities*. The connected hardware components are now displayed. Make note of the SCSI ID of your DVD drive.
3. Exit the menu with **Esc**.
4. Open *Configure Adapter Settings*. Under *Additional Options*, select *Boot Device Options* and press **Enter**.
5. Enter the ID of the DVD drive and press **Enter** again.
6. Press **Esc** twice to return to the start screen of the SCSI BIOS.
7. Exit this screen and confirm with *Yes* to boot the computer.

Regardless of what language and keyboard layout your final installation will be using, most BIOS configurations use the US keyboard layout as depicted in the following figure:



FIGURE 38.2: US KEYBOARD LAYOUT

38.2.4 Fails to Boot

Some hardware types, mainly very old or very recent ones, fail to install. Often this may happen because support for this type of hardware is missing in the installation kernel, or because of certain functionality included in this kernel, such as ACPI, that can still cause problems on some hardware.

If your system fails to install using the standard *Installation* mode from the first installation boot screen, try the following:

1. With the DVD still in the drive, reboot the machine with `Ctrl-Alt-Del` or using the hardware reset button.
2. When the boot screen appears, press `F5`, use the arrow keys of your keyboard to navigate to *No ACPI* and press `Enter` to launch the boot and installation process. This option disables the support for ACPI power management techniques.
3. Proceed with the installation as described in Book “Deployment Guide”, Chapter 6 “Installation with YaST”.

If this fails, proceed as above, but choose *Safe Settings* instead. This option disables ACPI and DMA support. Most hardware will boot with this option.

If both of these options fail, use the boot options prompt to pass any additional parameters needed to support this type of hardware to the installation kernel. For more information about the parameters available as boot options, refer to the kernel documentation located in </usr/src/linux/Documentation/kernel-parameters.txt>.



Tip: Obtaining Kernel Documentation

Install the `kernel-source` package to view the kernel documentation.

There are various other ACPI-related kernel parameters that can be entered at the boot prompt prior to booting for installation:

acpi=off

This parameter disables the complete ACPI subsystem on your computer. This may be useful if your computer cannot handle ACPI or if you think ACPI in your computer causes trouble.

acpi=force

Always enable ACPI even if your computer has an old BIOS dated before the year 2000. This parameter also enables ACPI if it is set in addition to acpi=off.

acpi=noirq

Do not use ACPI for IRQ routing.

acpi=ht

Run only enough ACPI to enable hyper-threading.

acpi=strict

Be less tolerant of platforms that are not strictly ACPI specification compliant.

pci=noacpi

Disable PCI IRQ routing of the new ACPI system.

pnpacpi=off

This option is for serial or parallel problems when your BIOS setup contains wrong interrupts or ports.

notsc

Disable the time stamp counter. This option can be used to work around timing problems on your systems. It is a recent feature, if you see regressions on your machine, especially time related or even total hangs, this option is worth a try.

nohz=off

Disable the nohz feature. If your machine hangs, this option may help. Otherwise it is of no use.

Once you have determined the right parameter combination, YaST automatically writes them to the boot loader configuration to make sure that the system boots properly next time.

If unexplainable errors occur when the kernel is loaded or during the installation, select *Memory Test* in the boot menu to check the memory. If *Memory Test* returns an error, it is usually a hardware error.

38.2.5 Fails to Launch Graphical Installer

After you insert the medium into your drive and reboot your machine, the installation screen comes up, but after you select *Installation*, the graphical installer does not start.

There are several ways to deal with this situation:

- Try to select another screen resolution for the installation dialogs.
- Select *Text Mode* for installation.
- Do a remote installation via VNC using the graphical installer.

PROCEDURE 38.3: CHANGE SCREEN RESOLUTION FOR INSTALLATION

1. Boot for installation.
2. Press **F3** to open a menu from which to select a lower resolution for installation purposes.

3. Select *Installation* and proceed with the installation as described in *Book “Deployment Guide”, Chapter 6 “Installation with YaST”*.

PROCEDURE 38.4: INSTALLATION IN TEXT MODE

1. Boot for installation.
2. Press **F3** and select *Text Mode*.
3. Select *Installation* and proceed with the installation as described in *Book “Deployment Guide”, Chapter 6 “Installation with YaST”*.

PROCEDURE 38.5: VNC INSTALLATION

1. Boot for installation.
2. Enter the following text at the boot options prompt:

```
vnc=1 vncpassword=some_password
```

Replace some_password with the password to use for VNC installation.

3. Select *Installation* then press **Enter** to start the installation.
Instead of starting right into the graphical installation routine, the system continues to run in a text mode, then halts, displaying a message containing the IP address and port number at which the installer can be reached via a browser interface or a VNC viewer application.
4. If using a browser to access the installer, launch the browser and enter the address information provided by the installation routines on the future SUSE Linux Enterprise Server machine and press **Enter**:

```
http://ip_address_of_machine:5801
```

A dialog opens in the browser window prompting you for the VNC password. Enter it and proceed with the installation as described in *Book “Deployment Guide”, Chapter 6 “Installation with YaST”*.



Important: Cross-platform Support

Installation via VNC works with any browser under any operating system, provided Java support is enabled.

Provide the IP address and password to your VNC viewer when prompted. A window opens, displaying the installation dialogs. Proceed with the installation as usual.

38.2.6 Only Minimalistic Boot Screen Started

You inserted the medium into the drive, the BIOS routines are finished, but the system does not start with the graphical boot screen. Instead it launches a very minimalistic text-based interface. This may happen on any machine not providing sufficient graphics memory for rendering a graphical boot screen.

Although the text boot screen looks minimalistic, it provides nearly the same functionality as the graphical one:

Boot Options

Unlike the graphical interface, the different boot options cannot be selected using the cursor keys of your keyboard. The boot menu of the text mode boot screen offers some keywords to enter at the boot prompt. These keywords map to the options offered in the graphical version. Enter your choice and press to launch the boot process.

Custom Boot Options

After selecting a boot option, enter the appropriate keyword at the boot prompt or enter some custom boot options as described in [Section 38.2.4, “Fails to Boot”](#). To launch the installation process, press .

Screen Resolutions

Use the F keys to determine the screen resolution for installation. If you need to boot in text mode, choose .

38.2.7 Log Files

For more information about log files that are created during installation, see [Section 37.5, “Gathering Information During the Installation”](#).

38.3 Boot Problems

Boot problems are situations when your system does not boot properly (does not boot to the expected target and login screen).

38.3.1 Fails to Load the GRUB 2 Boot Loader

If the hardware is functioning properly, it is possible that the boot loader is corrupted and Linux cannot start on the machine. In this case, it is necessary to repair the boot loader. To do so, you need to start the Rescue System as described in [Section 38.6.2, “Using the Rescue System”](#) and follow the instructions in [Section 38.6.2.4, “Modifying and Re-installing the Boot Loader”](#).

Other reasons for the machine not booting may be BIOS-related:

BIOS Settings

Check your BIOS for references to your hard disk. GRUB 2 may simply not be started if the hard disk itself cannot be found with the current BIOS settings.

BIOS Boot Order

Check whether your system's boot order includes the hard disk. If the hard disk option was not enabled, your system may install properly, but fails to boot when access to the hard disk is required.

38.3.2 No Graphical Login

If the machine comes up, but does not boot into the graphical login manager, anticipate problems either with the choice of the default systemd target or the configuration of the X Window System. To check the current systemd default target run the command **`sudo systemctl get-default`**. If the value returned is *not* `graphical.target`, run the command **`sudo systemctl isolate graphical.target`**. If the graphical login screen starts, log in and start *YaST > System > Services Manager* and set the *Default System Target* to *Graphical Interface*. From now on the system should boot into the graphical login screen.

If the graphical login screen does not start even if having booted or switched to the graphical target, your desktop or X Window software is probably misconfigured or corrupted. Examine the log files at `/var/log/Xorg.*.log` for detailed messages from the X server as it attempted to start. If the desktop fails during start, it may log error messages to the system journal that

can be queried with the command **journalctl** (see *Chapter 14, journalctl: Query the systemd Journal* for more information). If these error messages hint at a configuration problem in the X server, try to fix these issues. If the graphical system still does not come up, consider reinstalling the graphical desktop.

38.3.3 Root Btrfs Partition Cannot Be Mounted

If a **btrfs** root partition becomes corrupted, try the following options:

- Mount the partition with the **-o recovery** option.
- If that fails, run **btrfs-zero-log** on your root partition.

38.3.4 Force Checking Root Partitions

If the root partition becomes corrupted, use the parameter **forcefsck** on the boot prompt. This passes the option **-f** (force) to the **fsck** command.

38.4 Login Problems

Login problems are those where your machine does, in fact, boot to the expected welcome screen or login prompt, but refuses to accept the user name and password, or accepts them but then does not behave properly (fails to start the graphic desktop, produces errors, drops to a command line, etc.).

38.4.1 Valid User Name and Password Combinations Fail

This usually occurs when the system is configured to use network authentication or directory services and, for some reason, cannot retrieve results from its configured servers. The root user, as the only local user, is the only user that can still log in to these machines. The following are some common reasons a machine appears functional but cannot process logins correctly:

- The network is not working. For further directions on this, turn to [Section 38.5, “Network Problems”](#).
- DNS is not working at the moment (which prevents GNOME from working and the system from making validated requests to secure servers). One indication that this is the case is that the machine takes an extremely long time to respond to any action. Find more information about this topic in [Section 38.5, “Network Problems”](#).
- If the system is configured to use Kerberos, the system's local time may have drifted past the accepted variance with the Kerberos server time (this is typically 300 seconds). If NTP (network time protocol) is not working properly or local NTP servers are not working, Kerberos authentication ceases to function because it depends on common clock synchronization across the network.
- The system's authentication configuration is misconfigured. Check the PAM configuration files involved for any typographical errors or misordering of directives. For additional background information about PAM and the syntax of the configuration files involved, refer to *Book “Security Guide”, Chapter 2 “Authentication with PAM”*.
- The home partition is encrypted. Find more information about this topic in [Section 38.4.3, “Login to Encrypted Home Partition Fails”](#).

In all cases that do not involve external network problems, the solution is to reboot the system into single-user mode and repair the configuration before booting again into operating mode and attempting to log in again. To boot into single-user mode:

1. Reboot the system. The boot screen appears, offering a prompt.
2. Press `[Esc]` to exit the splash screen and get to the GRUB 2 text-based menu.
3. Press `[B]` to enter the GRUB 2 editor.
4. Add the following parameter to the line containing the Kernel parameters:

```
systemd.unit=rescue.target
```

5. Press `F10`.
6. Enter the user name and password for `root`.
7. Make all the necessary changes.
8. Boot into the full multiuser and network mode by entering `systemctl isolate graphical.target` at the command line.

38.4.2 Valid User Name and Password Not Accepted

This is by far the most common problem users encounter, because there are many reasons this can occur. Depending on whether you use local user management and authentication or network authentication, login failures occur for different reasons.

Local user management can fail for the following reasons:

- The user may have entered the wrong password.
- The user's home directory containing the desktop configuration files is corrupted or write protected.
- There may be problems with the X Window System authenticating this particular user, especially if the user's home directory has been used with another Linux distribution prior to installing the current one.

To locate the reason for a local login failure, proceed as follows:

1. Check whether the user remembered his password correctly before you start debugging the whole authentication mechanism. If the user may not remember his password correctly, use the YaST User Management module to change the user's password. Pay attention to the `Caps Lock` key and unlock it, if necessary.
2. Log in as `root` and check the system journal with `journalctl -e` for error messages of the login process and of PAM.
3. Try to log in from a console (using `Ctrl-Alt-F1`). If this is successful, the blame cannot be put on PAM, because it is possible to authenticate this user on this machine. Try to locate any problems with the X Window System or the GNOME desktop. For more information, refer to [Section 38.4.4, "Login Successful but GNOME Desktop Fails"](#).

4. If the user's home directory has been used with another Linux distribution, remove the `Xauthority` file in the user's home. Use a console login via `Ctrl-Alt-F1` and run `rm .Xauthority` as this user. This should eliminate X authentication problems for this user. Try graphical login again.
5. If the desktop could not start because of corrupt configuration files, proceed with *Section 38.4.4, "Login Successful but GNOME Desktop Fails"*.

In the following, common reasons a network authentication for a particular user may fail on a specific machine are listed:

- The user may have entered the wrong password.
- The user name exists in the machine's local authentication files and is also provided by a network authentication system, causing conflicts.
- The home directory exists but is corrupt or unavailable. Perhaps it is write protected or is on a server that is inaccessible at the moment.
- The user does not have permission to log in to that particular host in the authentication system.
- The machine has changed host names, for whatever reason, and the user does not have permission to log in to that host.
- The machine cannot reach the authentication server or directory server that contains that user's information.
- There may be problems with the X Window System authenticating this particular user, especially if the user's home has been used with another Linux distribution prior to installing the current one.

To locate the cause of the login failures with network authentication, proceed as follows:

1. Check whether the user remembered their password correctly before you start debugging the whole authentication mechanism.
2. Determine the directory server which the machine relies on for authentication and make sure that it is up and running and properly communicating with the other machines.
3. Determine that the user's user name and password work on other machines to make sure that his authentication data exists and is properly distributed.

4. See if another user can log in to the misbehaving machine. If another user can log in without difficulty or if root can log in, log in and examine the system journal with `journalctl -e > file`. Locate the time stamps that correspond to the login attempts and determine if PAM has produced any error messages.
5. Try to log in from a console (using `Ctrl-Alt-F1`). If this is successful, the problem is not with PAM or the directory server on which the user's home is hosted, because it is possible to authenticate this user on this machine. Try to locate any problems with the X Window System or the GNOME desktop. For more information, refer to [Section 38.4.4, "Login Successful but GNOME Desktop Fails"](#).
6. If the user's home directory has been used with another Linux distribution, remove the `Xauthority` file in the user's home. Use a console login via `Ctrl-Alt-F1` and run `rm .Xauthority` as this user. This should eliminate X authentication problems for this user. Try graphical login again.
7. If the desktop could not start because of corrupt configuration files, proceed with [Section 38.4.4, "Login Successful but GNOME Desktop Fails"](#).

38.4.3 Login to Encrypted Home Partition Fails

It is recommended to use an encrypted home partition for laptops. If you cannot log in to your laptop, the reason is usually simple: your partition could not be unlocked.

During the boot time, you need to enter the passphrase to unlock your encrypted partition. If you do not enter it, the boot process continues, leaving the partition locked.

To unlock your encrypted partition, proceed as follows:

1. Switch to the text console with `Ctrl-Alt-F1`.
2. Become root.
3. Restart the unlocking process again with:

```
systemctl restart home.mount
```
4. Enter your passphrase to unlock your encrypted partition.
5. Exit the text console and switch back to the login screen with `Alt-F7`.
6. Log in as usual.

38.4.4 Login Successful but GNOME Desktop Fails

If this is the case, it is likely that your GNOME configuration files have become corrupted. Some symptoms may include the keyboard failing to work, the screen geometry becoming distorted, or even the screen coming up as a bare gray field. The important distinction is that if another user logs in, the machine works normally. It is then likely that the problem can be fixed relatively quickly by simply moving the user's GNOME configuration directory to a new location, which causes GNOME to initialize a new one. Although the user is forced to reconfigure GNOME, no data is lost.

1. Switch to a text console by pressing `Ctrl`–`Alt`–`F1`.
2. Log in with your user name.
3. Move the user's GNOME configuration directories to a temporary location:

```
mv .gconf .gconf-ORIG-RECOVER
mv .gnome2 .gnome2-ORIG-RECOVER
```

4. Log out.
5. Log in again, but do not run any applications.
6. Recover your individual application configuration data (including the Evolution e-mail client data) by copying the `~/.gconf-ORIG-RECOVER/apps/` directory back into the new `~/.gconf` directory as follows:

```
cp -a .gconf-ORIG-RECOVER/apps .gconf/
```

If this causes the login problems, attempt to recover only the critical application data and reconfigure the remainder of the applications.

38.5 Network Problems

Many problems of your system may be network-related, even though they do not seem to be at first. For example, the reason for a system not allowing users to log in may be a network problem of some kind. This section introduces a simple checklist you can apply to identify the cause of any network problem encountered.

When checking the network connection of your machine, proceed as follows:

1. If you use an Ethernet connection, check the hardware first. Make sure that your network cable is properly plugged into your computer and router (or hub, etc.). The control lights next to your Ethernet connector are normally both be active.
If the connection fails, check whether your network cable works with another machine. If it does, your network card causes the failure. If hubs or switches are included in your network setup, they may be faulty, as well.
2. If using a wireless connection, check whether the wireless link can be established by other machines. If not, contact the wireless network's administrator.
3. Once you have checked your basic network connectivity, try to find out which service is not responding. Gather the address information of all network servers needed in your setup. Either look them up in the appropriate YaST module or ask your system administrator. The following list gives some typical network servers involved in a setup together with the symptoms of an outage.

DNS (Name Service)

A broken or malfunctioning name service affects the network's functionality in many ways. If the local machine relies on any network servers for authentication and these servers cannot be found because of name resolution issues, users would not even be able to log in. Machines in the network managed by a broken name server would not be able to “see” each other and communicate.

NTP (Time Service)

A malfunctioning or completely broken NTP service could affect Kerberos authentication and X server functionality.

NFS (File Service)

If any application needs data stored in an NFS mounted directory, it cannot start or function properly if this service was down or misconfigured. In the worst case scenario, a user's personal desktop configuration would not come up if their home directory containing the `.gconf` subdirectory could not be found because of a faulty NFS server.

Samba (File Service)

If any application needs data stored in a directory on a faulty Samba server, it cannot start or function properly.

NIS (User Management)

If your SUSE Linux Enterprise Server system relies on a faulty NIS server to provide the user data, users cannot log in to this machine.

LDAP (User Management)

If your SUSE Linux Enterprise Server system relies on a faulty LDAP server to provide the user data, users cannot log in to this machine.

Kerberos (Authentication)

Authentication will not work and login to any machine fails.

CUPS (Network Printing)

Users cannot print.

4. Check whether the network servers are running and whether your network setup allows you to establish a connection:



Important: Limitations

The debugging procedure described below only applies to a simple network server/client setup that does not involve any internal routing. It assumes both server and client are members of the same subnet without the need for additional routing.

- a. Use **ping** IP address or host name (replace host name with the host name of the server) to check whether each one of them is up and responding to the network. If this command is successful, it tells you that the host you were looking for is up and running and that the name service for your network is configured correctly. If ping fails with destination host unreachable, either your system or the desired server is not properly configured or down. Check whether your system is reachable by running **ping** IP address or your_host_name from another machine. If you can reach your machine from another machine, it is the server that is not running or not configured correctly.

If ping fails with `unknown host`, the name service is not configured correctly or the host name used was incorrect. For further checks on this matter, refer to [Step 4.b](#). If ping still fails, either your network card is not configured correctly or your network hardware is faulty.

- b. Use `host host name` to check whether the host name of the server you are trying to connect to is properly translated into an IP address and vice versa. If this command returns the IP address of this host, the name service is up and running. If the `host` command fails, check all network configuration files relating to name and address resolution on your host:

/etc/resolv.conf

This file is used to keep track of the name server and domain you are currently using. It can be modified manually or automatically adjusted by YaST or DHCP. Automatic adjustment is preferable. However, make sure that this file has the following structure and all network addresses and domain names are correct:

```
search fully_qualified_domain_name
nameserver ipaddress_of_nameserver
```

This file can contain more than one name server address, but at least one of them must be correct to provide name resolution to your host. If needed, adjust this file using the YaST Network Setting module (Hostname/DNS tab).

If your network connection is handled via DHCP, enable DHCP to change host name and name service information by selecting *Change Hostname via DHCP* and *Update Name Servers and Search List via DHCP* in the YaST DNS and Hostname module.

/etc/nsswitch.conf

This file tells Linux where to look for name service information. It should look like this:

```
...
hosts: files dns
networks: files dns
...
```

The `dns` entry is vital. It tells Linux to use an external name server. Normally, these entries are automatically managed by YaST, but it would be prudent to check.

If all the relevant entries on the host are correct, let your system administrator check the DNS server configuration for the correct zone information. For detailed information about DNS, refer to *Chapter 23, The Domain Name System*. If you have made sure that the DNS configuration of your host and the DNS server are correct, proceed with checking the configuration of your network and network device.

- c. If your system cannot establish a connection to a network server and you have excluded name service problems from the list of possible culprits, check the configuration of your network card.

Use the command `ip addr show network_device` to check whether this device was properly configured. Make sure that the `inet address` with the netmask (`/mask`) is configured correctly. An error in the IP address or a missing bit in your network mask would render your network configuration unusable. If necessary, perform this check on the server as well.

- d. If the name service and network hardware are properly configured and running, but some external network connections still get long time-outs or fail entirely, use `tracert fully_qualified_domain_name` (executed as `root`) to track the network route these requests are taking. This command lists any gateway (hop) that a request from your machine passes on its way to its destination. It lists the response time of each hop and whether this hop is reachable. Use a combination of `tracert` and `ping` to track down the culprit and let the administrators know.

Once you have identified the cause of your network trouble, you can resolve it yourself (if the problem is located on your machine) or let the system administrators of your network know about your findings so they can reconfigure the services or repair the necessary systems.

38.5.1 NetworkManager Problems

If you have a problem with network connectivity, narrow it down as described in *Procedure 38.6, "How to Identify Network Problems"*. If NetworkManager seems to be the culprit, proceed as follows to get logs providing hints on why NetworkManager fails:

1. Open a shell and log in as root.

2. Restart the NetworkManager:

```
systemctl restart NetworkManager
```

3. Open a Web page, for example, <http://www.opensuse.org> as normal user to see, if you can connect.

4. Collect any information about the state of NetworkManager in /var/log/NetworkManager.

For more information about NetworkManager, refer to *Chapter 34, Using NetworkManager*.

38.6 Data Problems

Data problems are when the machine may or may not boot properly but, in either case, it is clear that there is data corruption on the system and that the system needs to be recovered. These situations call for a backup of your critical data, enabling you to recover the system state from before your system failed. SUSE Linux Enterprise Server offers dedicated YaST modules for system backup and restoration and a rescue system that can be used to recover a corrupted system from the outside.

38.6.1 Managing Partition Images

Sometimes you need to perform a backup from an entire partition or even hard disk. Linux comes with the **dd** tool which can create an exact copy of your disk. Combined with **gzip** you save some space.

PROCEDURE 38.7: BACKING UP AND RESTORING HARD DISKS

1. Start a Shell as user root.

2. Select your source device. Typically this is something like /dev/sda (labeled as SOURCE).

3. Decide where you want to store your image (labeled as BACKUP_PATH). It must be different from your source device. In other words: if you make a backup from /dev/sda, your image file must not to be stored under /dev/sda.

4. Run the commands to create a compressed image file:

```
dd if=/dev/SOURCE | gzip > /BACKUP_PATH/image.gz
```

5. Restore the hard disk with the following commands:

```
gzip -dc /BACKUP_PATH/image.gz | dd of=/dev/SOURCE
```

If you only need to back up a partition, replace the `SOURCE` placeholder with your respective partition. In this case, your image file can lie on the same hard disk, but on a different partition.

38.6.2 Using the Rescue System

There are several reasons a system could fail to come up and run properly. A corrupted file system following a system crash, corrupted configuration files, or a corrupted boot loader configuration are the most common ones.

To help you to resolve these situations, SUSE Linux Enterprise Server contains a rescue system that you can boot. The rescue system is a small Linux system that can be loaded into a RAM disk and mounted as root file system, allowing you to access your Linux partitions from the outside. Using the rescue system, you can recover or modify any important aspect of your system.

- Manipulate any type of configuration file.
- Check the file system for defects and start automatic repair processes.
- Access the installed system in a “change root” environment.
- Check, modify, and re-install the boot loader configuration.
- Recover from a badly installed device driver or unusable kernel.
- Resize partitions using the parted command. Find more information about this tool at the GNU Parted Web site <http://www.gnu.org/software/parted/parted.html>.

The rescue system can be loaded from various sources and locations. The simplest option is to boot the rescue system from the original installation medium.



Note: IBM z Systems Starting the Rescue System

On IBM z Systems the installation system can be used for rescue purposes. To start the rescue system follow the instructions in [Section 38.7, “IBM z Systems: Using initrd as a Rescue System”](#).

1. Insert the installation medium into your DVD drive.
2. Reboot the system.
3. At the boot screen, press **F4** and choose *DVD-ROM*. Then choose *Rescue System* from the main menu.
4. Enter root at the Rescue: prompt. A password is not required.

If your hardware setup does not include a DVD drive, you can boot the rescue system from a network source. The following example applies to a remote boot scenario—if using another boot medium, such as a DVD, modify the info file accordingly and boot as you would for a normal installation.

1. Enter the configuration of your PXE boot setup and add the lines install=protocol://instsource and rescue=1. If you need to start the repair system, use repair=1 instead. As with a normal installation, protocol stands for any of the supported network protocols (NFS, HTTP, FTP, etc.) and instsource for the path to your network installation source.
2. Boot the system using “Wake on LAN”, as described in *Book “Deployment Guide”, Chapter 8 “Preparing the Boot of the Target System”, Section 8.7 “Wake on LAN”*.
3. Enter root at the Rescue: prompt. A password is not required.

Once you have entered the rescue system, you can use the virtual consoles that can be reached with **Alt-F1** to **Alt-F6**.

A shell and many other useful utilities, such as the mount program, are available in the /bin directory. The /sbin directory contains important file and network utilities for reviewing and repairing the file system. This directory also contains the most important binaries for system maintenance, such as fdisk, mkfs, mkswap, mount, and shutdown, ip and ss for maintaining the network. The directory /usr/bin contains the vi editor, find, less, and SSH.

To see the system messages, either use the command **dmesg** or view the system log with **journalctl**.

38.6.2.1 Checking and Manipulating Configuration Files

As an example for a configuration that might be fixed using the rescue system, imagine you have a broken configuration file that prevents the system from booting properly. You can fix this using the rescue system.

To manipulate a configuration file, proceed as follows:

1. Start the rescue system using one of the methods described above.
2. To mount a root file system located under /dev/sda6 to the rescue system, use the following command:

```
mount /dev/sda6 /mnt
```

All directories of the system are now located under /mnt

3. Change the directory to the mounted root file system:

```
cd /mnt
```

4. Open the problematic configuration file in the vi editor. Adjust and save the configuration.
5. Unmount the root file system from the rescue system:

```
umount /mnt
```

6. Reboot the machine.

38.6.2.2 Repairing and Checking File Systems

Generally, file systems cannot be repaired on a running system. If you encounter serious problems, you may not even be able to mount your root file system and the system boot may end with a “kernel panic”. In this case, the only way is to repair the system from the outside. The system contains the utilities to check and repair the btrfs, ext2, ext3, ext4, reiserfs, xfs, dosfs, and vfat file systems. Look for the command **fsck**. FILESYSTEM, for example, if you need a file system check for btrfs, use **fsck.btrfs**.

38.6.2.3 Accessing the Installed System

If you need to access the installed system from the rescue system, you need to do this in a *change root* environment. For example, to modify the boot loader configuration, or to execute a hardware configuration utility.

To set up a change root environment based on the installed system, proceed as follows:

1. Run **lsblk** to check which node corresponds to the root partition. It is /dev/sda2 in our example:

```
lsblk
NAME            MAJ:MIN RM   SIZE RO TYPE  MOUNTPOINT
sda              8:0    0 149,1G  0 disk
├─sda1           8:1    0    2G  0 part  [SWAP]
├─sda2           8:2    0   20G  0 part  /
└─sda3           8:3    0  127G  0 part
   └─cr_home     254:0    0  127G  0 crypt /home
```

2. Mount the root partition from the installed system:

```
mount /dev/sda2 /mnt
```

3. Mount /proc, /dev, and /sys partitions:

```
mount -t proc none /mnt/proc
mount --rbind /dev /mnt/dev
mount --rbind /sys /mnt/sys
```

4. Now you can “change root” into the new environment, keeping the bash shell:

```
chroot /mnt /bin/bash
```

5. Finally, mount the remaining partitions from the installed system:

```
mount -a
```

6. Now you have access to the installed system. Before rebooting the system, unmount the partitions with umount -a and leave the “change root” environment with exit.



Warning: Limitations

Although you have full access to the files and applications of the installed system, there are some limitations. The kernel that is running is the one that was booted with the rescue system, not with the change root environment. It only supports essential hardware and it is not possible to add kernel modules from the installed system unless the kernel versions are identical. Always check the version of the currently running (rescue) kernel with `uname -r` and then find out if a matching subdirectory exists in the `/lib/modules` directory in the change root environment. If yes, you can use the installed modules, otherwise you need to supply their correct versions on other media, such as a flash disk. Most often the rescue kernel version differs from the installed one — then you cannot simply access a sound card, for example. It is also not possible to start a graphical user interface.

Also note that you leave the “change root” environment when you switch the console with `Alt-F1` to `Alt-F6`.

38.6.2.4 Modifying and Re-installing the Boot Loader

Sometimes a system cannot boot because the boot loader configuration is corrupted. The start-up routines cannot, for example, translate physical drives to the actual locations in the Linux file system without a working boot loader.

To check the boot loader configuration and re-install the boot loader, proceed as follows:

1. Perform the necessary steps to access the installed system as described in [Section 38.6.2.3, “Accessing the Installed System”](#).
2. Check that the GRUB 2 boot loader is installed on the system. If not, install the package `grub2` and run

```
grub2-install /dev/sda
```

3. Check whether the following files are correctly configured according to the GRUB 2 configuration principles outlined in [Chapter 11, The Boot Loader GRUB 2](#) and apply fixes if necessary.

- /etc/default/grub
- /boot/grub2/device.map (optional file, only present if created manually)
- /boot/grub2/grub.cfg (this file is generated, do not edit)
- /etc/sysconfig/bootloader

4. Re-install the boot loader using the following command sequence:

```
grub2-mkconfig -o /boot/grub2/grub.cfg
```

5. Unmount the partitions, log out from the “change root” environment, and reboot the system:

```
umount -a  
exit  
reboot
```

38.6.2.5 Fixing Kernel Installation

A kernel update may introduce a new bug which can impact the operation of your system. For example a driver for a piece of hardware in your system may be faulty, which prevents you from accessing and using it. In this case, revert to the last working kernel (if available on the system) or install the original kernel from the installation media.



Tip: How to Keep Last Kernels after Update

To prevent failures to boot after a faulty kernel update, use the kernel multiversion feature and tell `libzypp` which kernels you want to keep after the update.

For example to always keep the last two kernels and the currently running one, add

```
multiversion.kernels = latest,latest-1,running
```

to the `/etc/zypp/zypp.conf` file. See *Book “Deployment Guide”, Chapter 14 “Installing Multiple Kernel Versions”* for more information.

A similar case is when you need to re-install or update a broken driver for a device not supported by SUSE Linux Enterprise Server. For example when a hardware vendor uses a specific device, such as a hardware RAID controller, which needs a binary driver to be recognized by the operating system. The vendor typically releases a Driver Update Disk (DUD) with the fixed or updated version of the required driver.

In both cases you need to access the installed system in the rescue mode and fix the kernel related problem, otherwise the system may fail to boot correctly:

1. Boot from the SUSE Linux Enterprise Server installation media.
2. If you are recovering after a faulty kernel update, skip this step. If you need to use a driver update disk (DUD), press **F6** to load the driver update after the boot menu appears, and choose the path or URL to the driver update and confirm with *Yes*.
3. Choose *Rescue System* from the boot menu and press **Enter**. If you chose to use DUD, you will be asked to specify where the driver update is stored.
4. Enter root at the Rescue: prompt. A password is not required.
5. Manually mount the target system and “change root” into the new environment. For more information, see [Section 38.6.2.3, “Accessing the Installed System”](#).
6. If using DUD, install/re-install/update the faulty device driver package. Always make sure the installed kernel version exactly matches the version of the driver you are installing. If fixing faulty kernel update installation, you can install the original kernel from the installation media with the following procedure.
 - a. Identify your DVD device with **hwinfo --cdrom** and mount it with **mount /dev/sr0 /mnt**.
 - b. Navigate to the directory where your kernel files are stored on the DVD, for example **cd /mnt/suse/x86_64/**.
 - c. Install required **kernel-***, **kernel-*-base**, and **kernel-*-extra** packages of your flavor with the **rpm -i** command.
7. Update configuration files and reinitialize the boot loader if needed. For more information, see [Section 38.6.2.4, “Modifying and Re-installing the Boot Loader”](#).
8. Remove any bootable media from the system drive and reboot.

38.7 IBM z Systems: Using initrd as a Rescue System

If the kernel of the SUSE® Linux Enterprise Server for IBM z Systems is upgraded or modified, it is possible to reboot the system accidentally in an inconsistent state, so standard procedures of IPLing the installed system fail. In such a case, you may use the installation system for rescue purposes.

IPL the SUSE Linux Enterprise Server for IBM z Systems installation system as described in Book “Deployment Guide”, Chapter 4 “Installation on IBM z Systems”, Section 4.2 “Preparing for Installation”. Choose *Start Installation* and enter all required parameters. After the installation system has loaded and you are asked which display type to use to control the installation, select SSH. Now you can log in to the system with SSH as root without a password.

In this state, no disks are configured. You need to configure them before you can proceed.

PROCEDURE 38.8: CONFIGURING DASDS

1. Configure DASDs with the following command:

```
dasd_configure 0.0.0150 1 0
```

0.0.0150 is the channel to which the DASD is connected. The 1 means activate the disk (a 0 at this place would deactivate the disk). The 0 stands for “no DIAG mode” for the disk (a 1 here would enable DAIG access to the disk).

2. Now the DASD is online (check with cat /proc/partitions) and can be used for subsequent commands.

PROCEDURE 38.9: CONFIGURING A ZFCP DISK

1. To configure a zFCP disk, it is necessary to first configure the zFCP adapter. Do this with the following command:

```
zfcphost_configure 0.0.4000 1
```

0.0.4000 is the channel to which the adapter is attached and 1 stands for activate (a 0 here would deactivate the adapter).

2. After the adapter is activated, a disk can be configured. Do this with the following command:

```
zfcplib_configure 0.0.4000 1234567887654321 8765432100000000 1
```

0.0.4000 is the previously-used channel ID, 1234567887654321 is the WWPN (World wide Port Number), and 8765432100000000 is the LUN (logical unit number). The 1 stands for activating the disk (a 0 here would deactivate the disk).

3. Now the zFCP disk is online (check with `cat /proc/partitions`) and can be used for subsequent commands.

Now the rescue system is fully set up and you can start repairing the installed system. See [Section 38.6.2, "Using the Rescue System"](#) for instructions on how to repair the most common issues.

A Documentation Updates

This chapter lists content changes for this document.

This manual was updated on the following dates:

- *Section A.1, "November 2016 (Initial Release of SUSE Linux Enterprise Server 12 SP2)"*
- *Section A.2, "March 2016 (Maintenance Release of SUSE Linux Enterprise Server 12 SP1)"*
- *Section A.3, "December 2015 (Initial Release of SUSE Linux Enterprise Server 12 SP1)"*
- *Section A.4, "February 2015 (Documentation Maintenance Update)"*
- *Section A.5, "October 2014 (Initial Release of SUSE Linux Enterprise Server 12)"*

A.1 November 2016 (Initial Release of SUSE Linux Enterprise Server 12 SP2)

General

- The e-mail address for documentation feedback has changed to doc-team@suse.com.
- The documentation for Docker has been enhanced and renamed to *Docker Guide*.

Chapter 3, YaST Online Update

- Mentioned in *Section 3.3, “Automatic Online Update”* that the automatic online update does not automatically restart the system afterward (Doc Comment #30116).

Chapter 5, Managing Software with Command Line Tools

- **zypper patch** no longer installs optional patches by default. To install optional patches, use the `--with-optional` parameter (FATE#320447).

Chapter 6, System Recovery and Snapshot Management with Snapper

- Added `/var/cache` and `/var/lib/libvirt/images` to *Section 6.1.2, “Directories That Are Excluded from Snapshots”* (Fate #320834).
- Added *Section 6.6, “Automatic Snapshot Clean-Up”*, which also includes documentation on Snapper's new quota support (Fate #312751).
- Added `Q:` (Fate #318799).

Chapter 10, Booting a Linux System

- Advised users to repair file system in case root file system fails on boot time (FATE#320443).

Chapter 11, The Boot Loader GRUB 2

- Added a hint on **grub-once** support of `/boot/grub2/custom.cfg` to *Section 11.2, “Configuration File Structure”* (Fate #319632).
- Added *Section 6.3.1, “Accessing and Identifying Snapshot Boot Entries”* (Fate #317972 and #318101).
- Added information about trusted boot support to *Section 11.3.3.3, “Boot Code Options Tab”* (Fate #316553).

Chapter 15, Basic Networking

- Added section about Network Teaming (FATE#320468), see *Section 15.8, “Setting Up Team Devices for Network Teaming”*.
- Mentioned `TUNNEL_DEVICE` for Wicked (FATE#317977, *Section 15.5.1.5, “Using Tunnels with Wicked”*).

Chapter 22, Time Synchronization with NTP

- Added information on the *Synchronize without Daemon* start-up option. Chroot jail is no longer the default (FATE #320392).

Note: NFSv2

- Added a note about enabling NFSv2 (https://bugzilla.suse.com/show_bug.cgi?id=919708 ↗).

Chapter 31, The Proxy Server Squid

- Updated chapter for Squid 3.5 (FATE#319674).

Section 37.5, "Gathering Information During the Installation"

- Added section about log files created during installation (FATE#320015).

Bugfixes

- Wrong service names for NFS with Kerberos (https://bugzilla.suse.com/show_bug.cgi?id=983230 ↗).
- Live patches are released based on the SUSE CVSS score (https://bugzilla.suse.com/show_bug.cgi?id=992101 ↗).

A.2 March 2016 (Maintenance Release of SUSE Linux Enterprise Server 12 SP1)

Chapter 10, Booting a Linux System

Added a note about initramfs migration from swap to LVM (https://bugzilla.suse.com/show_bug.cgi?id=992101 ↗)

A.3 December 2015 (Initial Release of SUSE Linux Enterprise Server 12 SP1)

General

- Book “*Subscription Management Tool for SLES 12 SP2*” is now part of the documentation for SUSE Linux Enterprise Server.
- Add-ons provided by SUSE have been renamed as modules and extensions. The manuals have been updated to reflect this change.
- Numerous small fixes and additions to the documentation, based on technical feedback.
- The registration service has been changed from Novell Customer Center to SUSE Customer Center.
- In YaST, you will now reach *Network Settings* via the *System* group. *Network Devices* is gone (https://bugzilla.suse.com/show_bug.cgi?id=867809 ↗).

Chapter 6, System Recovery and Snapshot Management with Snapper

- Added information about the new `--sync` switch for **snapper delete** to *Section 6.5.4, “Deleting Snapshots”* (Fate #317066).
- Added *Section 6.3.1, “Accessing and Identifying Snapshot Boot Entries”* (Fate #317972 and Fate #318101).
- Added a tip to *Section 6.3, “System Rollback by Booting from Snapshots”* on how to do a rollback to the initial installation state or the state before a system update (Fate #317973 and Fate #317900).
- Added *Section 6.1.3.3, “Creating and Mounting New Subvolumes”* (Fate #318805, https://bugzilla.suse.com/show_bug.cgi?id=910602 ↗).

Chapter 7, Remote Access with VNC

- Turned a note into a section, added information about VNC using secured protocol by default (Fate #318936), and removed `tightvnc` as it is fully replaced by `tigervnc`. All in *Section 7.2.1, “Available Configurations”*.

Chapter 5, Managing Software with Command Line Tools

- Added *Section 5.1.4, “Identifying Processes and Services Using Deleted Files”* (Fate #318827).
- Added more examples of `zypper list-patches --cve` in *Section 5.1.3.1, “Installing All Needed Patches”* (Fate #319053).
- Added *Section 5.1.2.6, “Installing Packages from Disabled Repositories”* and a tip on removing all `debuginfo` packages in *Section 5.1.2, “Installing and Removing Software with Zypper”* (Fate #316287).
- Added a sentence on announcing the need for the system reboot after specific patch is applied. (Fate #317872).

Chapter 14, journalctl: Query the systemd Journal

- Added section *Section 14.6, “Using YaST to Filter the systemd Journal”* (Fate #318486).

Chapter 11, The Boot Loader GRUB 2

- Updated/simplified the whole chapter to match the latest GRUB version, both command line and YaST version.

Chapter 12, UEFI (Unified Extensible Firmware Interface)

- Added *Section 12.1.4, “Using Non-Inbox Drivers”* (Fate #317593).

Chapter 15, Basic Networking

- Nanny is now on by default in *Section 15.5.1.3, “Nanny”* (Fate #318977).
- Added *Section 15.6, “Basic Router Setup”* (Fate #317121, https://bugzilla.suse.com/show_bug.cgi?id=870132).
- Added *Section 15.9, “Software-Defined Networking with Open vSwitch”* (Fate #318497).

Chapter 23, The Domain Name System

- Added *Section 23.3.2.9.1, “Adding Reverse Zones”* (Doc Comment #1356).

Chapter 25, Sharing File Systems with NFS

- Added tip saying that `--bind` mount is no longer necessary for NFSv4 mounts in *Section 25.3.2, “Exporting File Systems Manually”* (Fate #316311).

Section 8.1, “Available Data Synchronization Software”

- Mentioned cloud computing for file synchronization.

Chapter 29, The Apache HTTP Server

- Replaced `CA.sh` with the explicit `openssl` command in *Section 29.6.1.3, “Getting an Officially Signed Certificate”* (Doc Comment #28367).
- Added *Section 29.7, “Running Multiple Apache Instances on the Same Server”* (Fate #317786).
- Updated the chapter to match the latest Apache version 2.4 (Fate #319012).




Chapter 38, Common Problems and Their Solutions









- Improved GRUB 2 re-installation procedure in *Section 38.6.2.4, “Modifying and Re-installing the Boot Loader”*.

Part II, “System”

- Added *Chapter 20, Live Patching the Linux Kernel Using kGraft* (Fate #313296 and Fate #313438).

Bugfixes

- Removed obsolete `acpid.service` (https://bugzilla.suse.com/show_bug.cgi?id=918655 .
- Fixed wrong title in *Chapter 20, Live Patching the Linux Kernel Using kGraft* (https://bugzilla.suse.com/show_bug.cgi?id=954250 .
- Fixed wrong path name in *Chapter 29, The Apache HTTP Server* (https://bugzilla.suse.com/show_bug.cgi?id=949395 .

- Added a paragraph on secure boot enabled by default in *Section 12.1.1, “Implementation on SUSE Linux Enterprise”* (https://bugzilla.suse.com/show_bug.cgi?id=879486 .
- Removed documentation about VNC view-only passwords in *Section 7.3, “Persistent VNC Sessions”* because they are not available in SUSE Linux Enterprise Server (https://bugzilla.suse.com/show_bug.cgi?id=941307 .
- Fixed procedure to access the installed system in a rescue mode in *Section 38.6.2.3, “Accessing the Installed System”* (https://bugzilla.suse.com/show_bug.cgi?id=918217 .
- Added a new tip on updating the initramfs file after changing the default **sysctl** configuration in *Section 10.2, “initramfs”* (https://bugzilla.suse.com/show_bug.cgi?id=927506 .
- Added a tip on preventing wicked from deactivating the network device on NFS roots in *Section 25.4.1, “Importing File Systems with YaST”* and *Section 15.4.1.2.5, “Activating the Network Device”* (https://bugzilla.suse.com/show_bug.cgi?id=938152 .
- Fixed misleading statement about `kernel-FLAVOR-extra` in *Section 37.6, “Support of Kernel Modules”* (http://bugzilla.suse.com/show_bug.cgi?id=922976 .
- Btrfs/Snapper: Snapshots with new Subvolumes will not be Deleted (https://bugzilla.suse.com/show_bug.cgi?id=910602 .
- Btrfs Documentation on Separate Subvolume on /var/lib and Supportability (https://bugzilla.suse.com/show_bug.cgi?id=930424 .



A.4 February 2015 (Documentation Maintenance Update)

Chapter 13, The systemd Daemon

A typo in a command has been fixed (https://bugzilla.suse.com/show_bug.cgi?id=900219 .

A.5 October 2014 (Initial Release of SUSE Linux Enterprise Server 12)

General

- Removed all KDE documentation and references because KDE is no longer shipped.
- Removed all references to SuSEconfig, which is no longer supported (Fate #100011).
- Move from System V init to systemd (Fate #310421). Updated affected parts of the documentation.
- YaST Runlevel Editor has changed to Services Manager (Fate #312568). Updated affected parts of the documentation.
- Removed all references to ISDN support, as ISDN support has been removed (Fate #314594).
- Removed all references to the YaST DSL module as it is no longer shipped (Fate #316264).
- Removed all references to the YaST Modem module as it is no longer shipped (Fate #316264).
- Btrfs has become the default file system for the root partition (Fate #315901). Updated affected parts of the documentation.
- The **dmesg** now provides human-readable time stamps in `ctime()`-like format (Fate #316056). Updated affected parts of the documentation.
- syslog and syslog-ng have been replaced by rsyslog (Fate #316175). Updated affected parts of the documentation.
- MariaDB is now shipped as the relational database instead of MySQL (Fate #313595). Updated affected parts of the documentation.
- SUSE-related products are no longer available from <http://download.novell.com>  but from <http://download.suse.com> . Adjusted links accordingly.
- Novell Customer Center has been replaced with SUSE Customer Center. Updated affected parts of the documentation.

- /var/run is mounted as tmpfs (Fate #303793). Updated affected parts of the documentation.
- The following architectures are no longer supported: IA64 and x86. Updated affected parts of the documentation.
- The traditional method for setting up the network with ifconfig has been replaced by wicked. Updated affected parts of the documentation.
- A lot of networking commands are deprecated and have been replaced by newer commands (usually ip). Updated affected parts of the documentation.

arp: ip neighbor

ifconfig: ip addr, ip link

iptunnel: ip tunnel

iwconfig: iw

nameif: ip link, ifrename

netstat: ss, ip route, ip -s link, ip maddr


route: ip route

- Numerous small fixes and additions to the documentation, based on technical feedback.

Chapter 3, YaST Online Update

- YaST provides an option to enable or disable the use of delta RPMs (Fate #314867).
- Before installing patches that require a reboot, you are notified by YaST and can choose how to proceed.

Chapter 37, Gathering System Information for Support

- Added the section *Section 37.1, "Displaying Current System Information"* (Fate #315869).
- Added a section about the Supportconfig Analysis (SCA) tool and appliance: *Section 37.4, "Analyzing System Information"* (Fate #315699).
- Added the section *Section 37.6, "Support of Kernel Modules"* (http://bugzilla.suse.com/show_bug.cgi?id=869159 .
- Updated and restructured the chapter.

Chapter 4, YaST in Text Mode

- Added information on how to filter and select packages in the software installation module.

Chapter 6, System Recovery and Snapshot Management with Snapper

- Updated the chapter and added new features (Fate #312751, Fate #316238, Fate #316233, Fate #316232, Fate #316222, Fate #316203, Fate #316222).
- Added the section *Section 6.3, "System Rollback by Booting from Snapshots"* (Fate #316231, Fate #316221, Fate #316541, Fate #316522).

Chapter 7, Remote Access with VNC

- The default VNC viewer is now tigervnc.
- Added corrections on window manager start-up in persistent VNC sessions.

Chapter 5, Managing Software with Command Line Tools

- Removed documentation about Zypper's rug compatibility mode (Fate #317708).
- Rewrote *Section 5.1.6, "Querying Repositories and Packages with Zypper"*.

Chapter 10, Booting a Linux System

- Significantly shortened the chapter, because System V init has been replaced by systemd. systemd is now described in a separate chapter: *Chapter 13, The systemd Daemon*.

Chapter 13, The systemd Daemon

- Added a new chapter on systemd and the YaST Services Manager (Fate #316631, Fate #312568).
- New section on loading kernel modules (http://bugzilla.suse.com/show_bug.cgi?id=892349 ↗).

Chapter 14, journalctl: Query the systemd Journal

Added a new chapter (http://bugzilla.suse.com/show_bug.cgi?id=878352 ↗).

Chapter 11, The Boot Loader GRUB 2

- Replaced the GRUB Legacy documentation with a new chapter on GRUB 2.
- Support for LILO has been dropped.
- Added new section *Section 11.4, “Differences in Terminal Usage on z Systems”*.

Chapter 12, UEFI (Unified Extensible Firmware Interface)

- Updated the chapter and added new features (Fate #314510, Fate #316365).
- Added instructions on where to find the SUSE Key certificate (Doc Comment #25080).

Chapter 16, Printer Operation

Updated chapter and section according to new CUPS version and with PDF now being a common printing data format (Fate #314630).

Chapter 17, The X Window System

- Updated the chapter to reflect dynamic configuration during each start-up.
- Updated *Section 17.1, “Installing and Configuring Fonts”*.

Chapter 15, Basic Networking

- NetworkManager is now part of the Workstation Extension: *Section 15.4.1.1, “Configuring Global Networking Options”* (Fate #316888).
- Added section about new **wicked** framework for network configuration: *Section 15.5, “Configuring a Network Connection Manually”* (Fate #316649).
- Mentioned additional options that can be added to /etc/resolv.conf: *Section 15.5.2, “Configuration Files”* (Fate #316048).

Chapter 28, SLP

- Re-wrote chapter to provide significantly more information on the slptool command.

Chapter 23, The Domain Name System

- The YaST DNS module now supports configuring forwarders (Fate #309036).

Chapter 24, DHCP

- Removed dhcpcd as it is no longer shipped (Fate #316111).

Chapter 26, Samba

- Added section *Section 26.8, "Advanced Topics"*.
- Added section *Section 26.8.1, "Transparent File Compression on Btrfs"*.
- Added section *Section 26.8.2, "Snapshots"*.

Chapter 25, Sharing File Systems with NFS

- Configuring NFSv4 shares is now mostly similar to NFSv3, especially the previously required `bind mount` setting is now deprecated (Fate #315589).
- Mounting NFS volumes locally on the exporting server is not supported.

Chapter 27, On-Demand Mounting with Autofs

- Added a chapter on autofs (Fate #316185).

Chapter 29, The Apache HTTP Server

- Removed references to `mono` and `mod_mono` as it was removed from the distribution.
- Updated the chapter to Apache version 2.4 (Fate #316067).
- Removed deprecated directive NameVirtualHost and updated *Section 29.2.2.1, "Virtual Host Configuration"* accordingly.
- Updated Order, Allow, and Deny directives with standard Require.
- Removed virtual "Snake Oil" company from *Section 29.6, "Setting Up a Secure Web Server with SSL"*.

Chapter 30, Setting Up an FTP Server with YaST

- pure-ftpd was dropped (Fate #315176, Fate #316308).

Chapter 35, Power Management

- Removed obsolete references to the pm-utils package.

Chapter 38, Common Problems and Their Solutions

- Added new section *Section 38.3.3, "Root Btrfs Partition Cannot Be Mounted"* (Fate #308679, Fate #315126).
- Removed section about deprecated YaST Repair module (Fate #308679).

Wi-Fi Configuration

- Removed chapter about Wi-Fi configuration with YaST as Wi-Fi configuration can be done with NetworkManager: *Chapter 34, Using NetworkManager*.

Tablet PCs

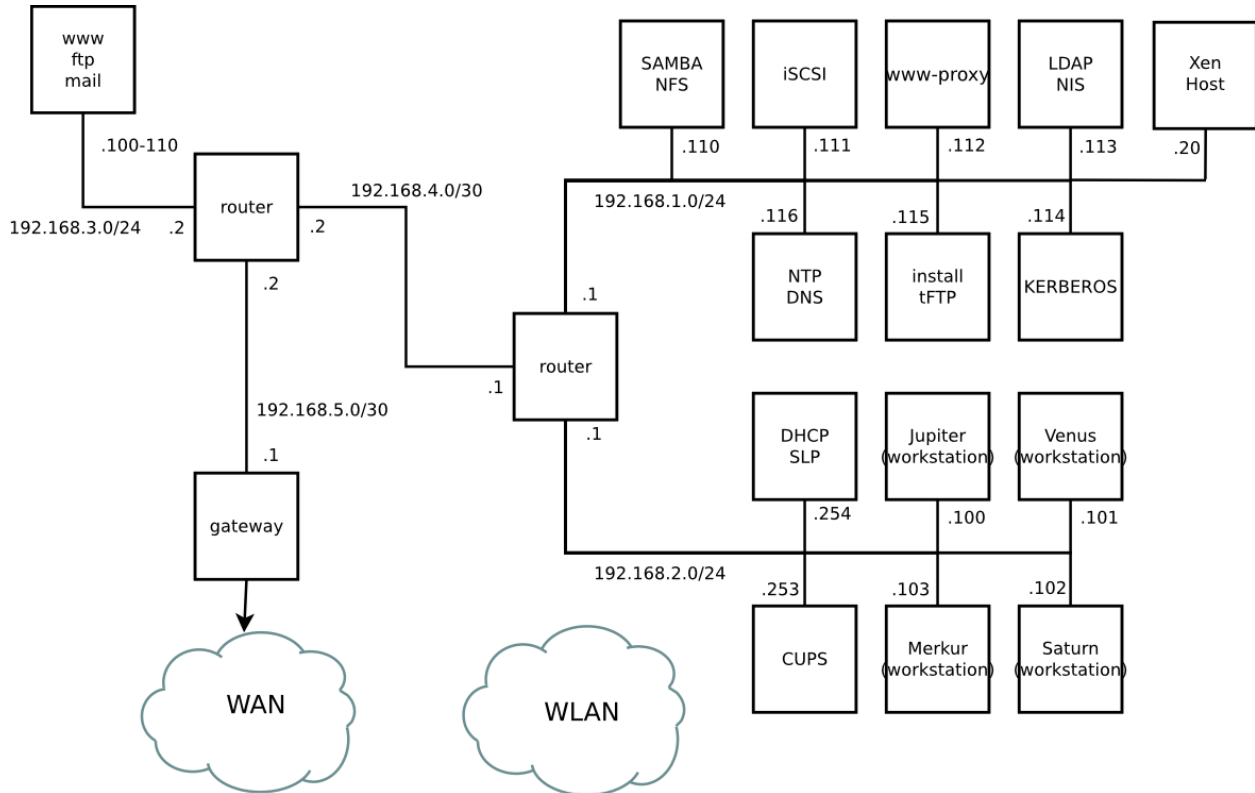
- Removed deprecated chapter about tablet PCs.

Bugfixes

- Added the section *Section 37.6, "Support of Kernel Modules"* (http://bugzilla.suse.com/show_bug.cgi?id=869159 ↗).
- Added a new chapter *Chapter 14, journalctl: Query the systemd Journal* (http://bugzilla.suse.com/show_bug.cgi?id=878352 ↗).

B An Example Network

This example network is used across all network-related chapters of the SUSE® Linux Enterprise Server documentation.



C GNU Licenses

This appendix contains the GNU Free Documentation License version 1.2.

GNU Free Documentation License

Copyright (C) 2000, 2001, 2002 Free Software Foundation, Inc. 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA. Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

0. PREAMBLE

The purpose of this License is to make a manual, textbook, or other functional and useful document "free" in the sense of freedom: to assure everyone the effective freedom to copy and redistribute it, with or without modifying it, either commercially or non-commercially. Secondly, this License preserves for the author and publisher a way to get credit for their work, while not being considered responsible for modifications made by others.

This License is a kind of "copyleft", which means that derivative works of the document must themselves be free in the same sense. It complements the GNU General Public License, which is a copyleft license designed for free software.

We have designed this License to use it for manuals for free software, because free software needs free documentation: a free program should come with manuals providing the same freedoms that the software does. But this License is not limited to software manuals; it can be used for any textual work, regardless of subject matter or whether it is published as a printed book. We recommend this License principally for works whose purpose is instruction or reference.

1. APPLICABILITY AND DEFINITIONS

This License applies to any manual or other work, in any medium, that contains a notice placed by the copyright holder saying it can be distributed under the terms of this License. Such a notice grants a world-wide, royalty-free license, unlimited in duration, to use that work under the conditions stated herein. The "Document", below, refers to any such manual or work. Any member of the public is a licensee, and is addressed as "you". You accept the license if you copy, modify or distribute the work in a way requiring permission under copyright law.

A "Modified Version" of the Document means any work containing the Document or a portion of it, either copied verbatim, or with modifications and/or translated into another language.

A "Secondary Section" is a named appendix or a front-matter section of the Document that deals exclusively with the relationship of the publishers or authors of the Document to the Document's overall subject (or to related matters) and contains nothing that could fall directly within that overall subject. (Thus, if the Document is in part a textbook of mathematics, a Secondary Section may not explain any mathematics.) The relationship could be a matter of historical connection with the subject or with related matters, or of legal, commercial, philosophical, ethical or political position regarding them.

The "Invariant Sections" are certain Secondary Sections whose titles are designated, as being those of Invariant Sections, in the notice that says that the Document is released under this License. If a section does not fit the above definition of Secondary then it is not allowed to be designated as Invariant. The Document may contain zero Invariant Sections. If the Document does not identify any Invariant Sections then there are none.

The "Cover Texts" are certain short passages of text that are listed, as Front-Cover Texts or Back-Cover Texts, in the notice that says that the Document is released under this License. A Front-Cover Text may be at most 5 words, and a Back-Cover Text may be at most 25 words.

A "Transparent" copy of the Document means a machine-readable copy, represented in a format whose specification is available to the general public, that is suitable for revising the document straightforwardly with generic text editors or (for images composed of pixels) generic paint programs or (for drawings) some widely available draw-

ing editor, and that is suitable for input to text formatters or for automatic translation to a variety of formats suitable for input to text formatters. A copy made in an otherwise Transparent file format whose markup, or absence of markup, has been arranged to thwart or discourage subsequent modification by readers is not Transparent. An image format is not Transparent if used for any substantial amount of text. A copy that is not "Transparent" is called "Opaque".

Examples of suitable formats for Transparent copies include plain ASCII without markup, Texinfo input format, LaTeX input format, SGML or XML using a publicly available DTD, and standard-conforming simple HTML, PostScript or PDF designed for human modification. Examples of transparent image formats include PNG, XCF and JPG. Opaque formats include proprietary formats that can be read and edited only by proprietary word processors, SGML or XML for which the DTD and/or processing tools are not generally available, and the machine-generated HTML, PostScript or PDF produced by some word processors for output purposes only.

The "Title Page" means, for a printed book, the title page itself, plus such following pages as are needed to hold, legibly, the material this License requires to appear in the title page. For works in formats which do not have any title page as such, "Title Page" means the text near the most prominent appearance of the work's title, preceding the beginning of the body of the text.

A section "Entitled XYZ" means a named subunit of the Document whose title either is precisely XYZ or contains XYZ in parentheses following text that translates XYZ in another language. (Here XYZ stands for a specific section name mentioned below, such as "Acknowledgements", "Dedications", "Endorsements", or "History".) To "Preserve the Title" of such a section when you modify the Document means that it remains a section "Entitled XYZ" according to this definition.

The Document may include Warranty Disclaimers next to the notice which states that this License applies to the Document. These Warranty Disclaimers are considered to be included by reference in this License, but only as regards disclaiming warranties: any other implication that these Warranty Disclaimers may have is void and has no effect on the meaning of this License.

2. VERBATIM COPYING

You may copy and distribute the Document in any medium, either commercially or non-commercially, provided that this License, the copyright notices, and the license notice saying this License applies to the Document are reproduced in all copies, and that you add no other conditions whatsoever to those of this License. You may not use technical measures to obstruct or control the reading or further copying of the copies you make or distribute. However, you may accept compensation in exchange for copies. If you distribute a large enough number of copies you must also follow the conditions in section 3.

You may also lend copies, under the same conditions stated above, and you may publicly display copies.

3. COPYING IN QUANTITY

If you publish printed copies (or copies in media that commonly have printed covers) of the Document, numbering more than 100, and the Document's license notice requires Cover Texts, you must enclose the copies in covers that carry, clearly and legibly, all these Cover Texts: Front-Cover Texts on the front cover, and Back-Cover Texts on the back cover. Both covers must also clearly and legibly identify you as the publisher of these copies. The front cover must present the full title with all words of the title equally prominent and visible. You may add other material on the covers in addition. Copying with changes limited to the covers, as long as they preserve the title of the Document and satisfy these conditions, can be treated as verbatim copying in other respects.

If the required texts for either cover are too voluminous to fit legibly, you should put the first ones listed (as many as fit reasonably) on the actual cover, and continue the rest onto adjacent pages.

If you publish or distribute Opaque copies of the Document numbering more than 100, you must either include a machine-readable Transparent copy along with each Opaque copy, or state in or with each Opaque copy a computer-network location from

which the general network-using public has access to download using public-standard network protocols a complete Transparent copy of the Document, free of added material. If you use the latter option, you must take reasonably prudent steps, when you begin distribution of Opaque copies in quantity, to ensure that this Transparent copy will remain thus accessible at the stated location until at least one year after the last time you distribute an Opaque copy (directly or through your agents or retailers) of that edition to the public.

It is requested, but not required, that you contact the authors of the Document well before redistributing any large number of copies, to give them a chance to provide you with an updated version of the Document.

4. MODIFICATIONS

You may copy and distribute a Modified Version of the Document under the conditions of sections 2 and 3 above, provided that you release the Modified Version under precisely this License, with the Modified Version filling the role of the Document, thus licensing distribution and modification of the Modified Version to whoever possesses a copy of it. In addition, you must do these things in the Modified Version:

- A. Use in the Title Page (and on the covers, if any) a title distinct from that of the Document, and from those of previous versions (which should, if there were any, be listed in the History section of the Document). You may use the same title as a previous version if the original publisher of that version gives permission.
- B. List on the Title Page, as authors, one or more persons or entities responsible for authorship of the modifications in the Modified Version, together with at least five of the principal authors of the Document (all of its principal authors, if it has fewer than five), unless they release you from this requirement.
- C. State on the Title page the name of the publisher of the Modified Version, as the publisher.
- D. Preserve all the copyright notices of the Document.
- E. Add an appropriate copyright notice for your modifications adjacent to the other copyright notices.
- F. Include, immediately after the copyright notices, a license notice giving the public permission to use the Modified Version under the terms of this License, in the form shown in the Addendum below.
- G. Preserve in that license notice the full lists of Invariant Sections and required Cover Texts given in the Document's license notice.
- H. Include an unaltered copy of this License.
- I. Preserve the section Entitled "History", Preserve its Title, and add to it an item stating at least the title, year, new authors, and publisher of the Modified Version as given on the Title Page. If there is no section Entitled "History" in the Document, create one stating the title, year, authors, and publisher of the Document as given on its Title Page, then add an item describing the Modified Version as stated in the previous sentence.
- J. Preserve the network location, if any, given in the Document for public access to a Transparent copy of the Document, and likewise the network locations given in the Document for previous versions it was based on. These may be placed in the "History" section. You may omit a network location for a work that was published at least four years before the Document itself, or if the original publisher of the version it refers to gives permission.
- K. For any section Entitled "Acknowledgements" or "Dedications", Preserve the Title of the section, and preserve in the section all the substance and tone of each of the contributor acknowledgements and/or dedications given therein.
- L. Preserve all the Invariant Sections of the Document, unaltered in their text and in their titles. Section numbers or the equivalent are not considered part of the section titles.
- M. Delete any section Entitled "Endorsements". Such a section may not be included in the Modified Version.

- N. Do not retitle any existing section to be Entitled "Endorsements" or to conflict in title with any Invariant Section.
- O. Preserve any Warranty Disclaimers.

If the Modified Version includes new front-matter sections or appendices that qualify as Secondary Sections and contain no material copied from the Document, you may at your option designate some or all of these sections as invariant. To do this, add their titles to the list of Invariant Sections in the Modified Version's license notice. These titles must be distinct from any other section titles.

You may add a section Entitled "Endorsements", provided it contains nothing but endorsements of your Modified Version by various parties—for example, statements of peer review or that the text has been approved by an organization as the authoritative definition of a standard.

You may add a passage of up to five words as a Front-Cover Text, and a passage of up to 25 words as a Back-Cover Text, to the end of the list of Cover Texts in the Modified Version. Only one passage of Front-Cover Text and one of Back-Cover Text may be added by (or through arrangements made by) any one entity. If the Document already includes a cover text for the same cover, previously added by you or by arrangement made by the same entity you are acting on behalf of, you may not add another; but you may replace the old one, on explicit permission from the previous publisher that added the old one.

The author(s) and publisher(s) of the Document do not by this License give permission to use their names for publicity for or to assert or imply endorsement of any Modified Version.

5. COMBINING DOCUMENTS

You may combine the Document with other documents released under this License, under the terms defined in section 4 above for modified versions, provided that you include in the combination all of the Invariant Sections of all of the original documents, unmodified, and list them all as Invariant Sections of your combined work in its license notice, and that you preserve all their Warranty Disclaimers.

The combined work need only contain one copy of this License, and multiple identical Invariant Sections may be replaced with a single copy. If there are multiple Invariant Sections with the same name but different contents, make the title of each such section unique by adding at the end of it, in parentheses, the name of the original author or publisher of that section if known, or else a unique number. Make the same adjustment to the section titles in the list of Invariant Sections in the license notice of the combined work.

In the combination, you must combine any sections Entitled "History" in the various original documents, forming one section Entitled "History"; likewise combine any sections Entitled "Acknowledgements", and any sections Entitled "Dedications". You must delete all sections Entitled "Endorsements".

6. COLLECTIONS OF DOCUMENTS

You may make a collection consisting of the Document and other documents released under this License, and replace the individual copies of this License in the various documents with a single copy that is included in the collection, provided that you follow the rules of this License for verbatim copying of each of the documents in all other respects.

You may extract a single document from such a collection, and distribute it individually under this License, provided you insert a copy of this License into the extracted document, and follow this License in all other respects regarding verbatim copying of that document.

7. AGGREGATION WITH INDEPENDENT WORKS

A compilation of the Document or its derivatives with other separate and independent documents or works, in or on a volume of a storage or distribution medium, is called an "aggregate" if the copyright resulting from the compilation is not used to limit the legal rights of the compilation's users beyond what the individual works permit. When the Document is included in an aggregate, this License does not apply to the other works in the aggregate which are not themselves derivative works of the Document.

If the Cover Text requirement of section 3 is applicable to these copies of the Document, then if the Document is less than one half of the entire aggregate, the Document's Cover Texts may be placed on covers that bracket the Document within the aggregate, or the electronic equivalent of covers if the Document is in electronic form. Otherwise they must appear on printed covers that bracket the whole aggregate.

8. TRANSLATION

Translation is considered a kind of modification, so you may distribute translations of the Document under the terms of section 4. Replacing Invariant Sections with translations requires special permission from their copyright holders, but you may include translations of some or all Invariant Sections in addition to the original versions of these Invariant Sections. You may include a translation of this License, and all the license notices in the Document, and any Warranty Disclaimers, provided that you also include the original English version of this License and the original versions of those notices and disclaimers. In case of a disagreement between the translation and the original version of this License or a notice or disclaimer, the original version will prevail.

If a section in the Document is Entitled "Acknowledgements", "Dedications", or "History", the requirement (section 4) to Preserve its Title (section 1) will typically require changing the actual title.

9. TERMINATION

You may not copy, modify, sublicense, or distribute the Document except as expressly provided for under this License. Any other attempt to copy, modify, sublicense or distribute the Document is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

10. FUTURE REVISIONS OF THIS LICENSE

The Free Software Foundation may publish new, revised versions of the GNU Free Documentation License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns. See <http://www.gnu.org/copyleft/>.

Each version of the License is given a distinguishing version number. If the Document specifies that a particular numbered version of this License "or any later version" applies to it, you have the option of following the terms and conditions either of that specified version or of any later version that has been published (not as a draft) by the Free Software Foundation. If the Document does not specify a version number of this License, you may choose any version ever published (not as a draft) by the Free Software Foundation.

ADDENDUM: How to use this License for your documents

```
Copyright (c) YEAR YOUR NAME.
Permission is granted to copy, distribute and/or modify this document
under the terms of the GNU Free Documentation License, Version 1.2
or any later version published by the Free Software Foundation;
with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts.
A copy of the license is included in the section entitled "GNU
Free Documentation License".
```

If you have Invariant Sections, Front-Cover Texts and Back-Cover Texts, replace the "with...Texts." line with this:

```
with the Invariant Sections being LIST THEIR TITLES, with the
Front-Cover Texts being LIST, and with the Back-Cover Texts being LIST.
```

If you have Invariant Sections without Cover Texts, or some other combination of the three, merge those two alternatives to suit the situation.

If your document contains nontrivial examples of program code, we recommend releasing these examples in parallel under your choice of free software license, such as the GNU General Public License, to permit their use in free software.